Amplifiers and Oscillators
53-210

Feedback

Feedback Instruments Ltd, Park Road, Crowborough, E. Sussex, TN6 2QR, UK.
Telephone: +44 (0) 1892 653322, Fax: +44 (0) 1892 663719.
email: feedback@feedback-instruments.com website: http://www.feedback-instruments.com

Manual produced from software version: v2.3

Date: 16/08/2011

Feedback Part No. 1160–53210

THE HEALTH AND SAFETY AT WORK ACT 1974

We are required under the Health and Safety at Work Act 1974, to make available to users of this equipment certain information regarding its safe use.+

The equipment, when used in normal or prescribed applications within the parameters set for its mechanical and electrical performance, should not cause any danger or hazard to health or safety if normal engineering practices are observed and they are used in accordance with the instructions supplied.

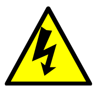
If, in specific cases, circumstances exist in which a potential hazard may be brought about by careless or improper use, these will be pointed out and the necessary precautions emphasised.

While we provide the fullest possible user information relating to the proper use of this equipment, if there is any doubt whatsoever about any aspect, the user should contact the Product Safety Officer at Feedback Instruments Limited, Crowborough.

This equipment should not be used by inexperienced users unless they are under supervision.

We are required by European Directives to indicate on our equipment panels certain areas and warnings that require attention by the user. These have been indicated in the specified way by yellow labels with black printing, the meaning of any labels that may be fixed to the instrument are shown below:

CAUTION -
RISK OF
DANGER



CAUTION -
RISK OF
ELECTRIC SHOCK

CAUTION -
ELECTROSTATIC
SENSITIVE DEVICE

Refer to accompanying documents

PRODUCT IMPROVEMENTS

We maintain a policy of continuous product improvement by incorporating the latest developments and components into our equipment, even up to the time of dispatch.

All major changes are incorporated into up-dated editions of our manuals and this manual was believed to be correct at the time of printing. However, some product changes which do not affect the instructional capability of the equipment, may not be included until it is necessary to incorporate other significant changes.

COMPONENT REPLACEMENT

Where components are of a ‘Safety Critical’ nature, i.e. all components involved with the supply or carrying of voltages at supply potential or higher, these must be replaced with components of equal international safety approval in order to maintain full equipment safety.

In order to maintain compliance with international directives, all replacement components should be identical to those originally supplied.

Any component may be ordered direct from Feedback or its agents by quoting the following information:

1. Equipment type	2. Component value
3. Component reference	4. Equipment serial number

Components can often be replaced by alternatives available locally, however we cannot therefore guarantee continued performance either to published specification or compliance with international standards.

OPERATING CONDITIONS

WARNING:

This equipment must not be used in conditions of condensing humidity.

This equipment is designed to operate under the following conditions:

Operating Temperature	10°C to 40°C (50°F to 104°F)
Humidity	10% to 90% (non-condensing)

CE DECLARATION CONCERNING ELECTROMAGNETIC COMPATIBILITY

Should this equipment be used outside the classroom, laboratory study area or similar such place for which it is designed and sold then Feedback Instruments Ltd hereby states that conformity with the protection requirements of the European Community Electromagnetic Compatibility Directive (89/336/EEC) may be invalidated and could lead to prosecution.

This equipment, when operated in accordance with the supplied documentation, does not cause electromagnetic disturbance outside its immediate electromagnetic environment.

ACKNOWLEDGEMENTS

Feedback Instruments Ltd acknowledge all trademarks.

IBM, IBM - PC are registered trademarks of International Business Machines.

MICROSOFT, WINDOWS 7, WINDOWS VISTA, WINDOWS XP, WINDOWS 2000, WINDOWS NT, WINDOWS ME, WINDOWS 98, WINDOWS 95, WINDOWS 3.1 and Internet Explorer are registered trademarks of Microsoft Corporation.

Familiarisation

Objectives

To become familiar with the circuit blocks available on the workboard

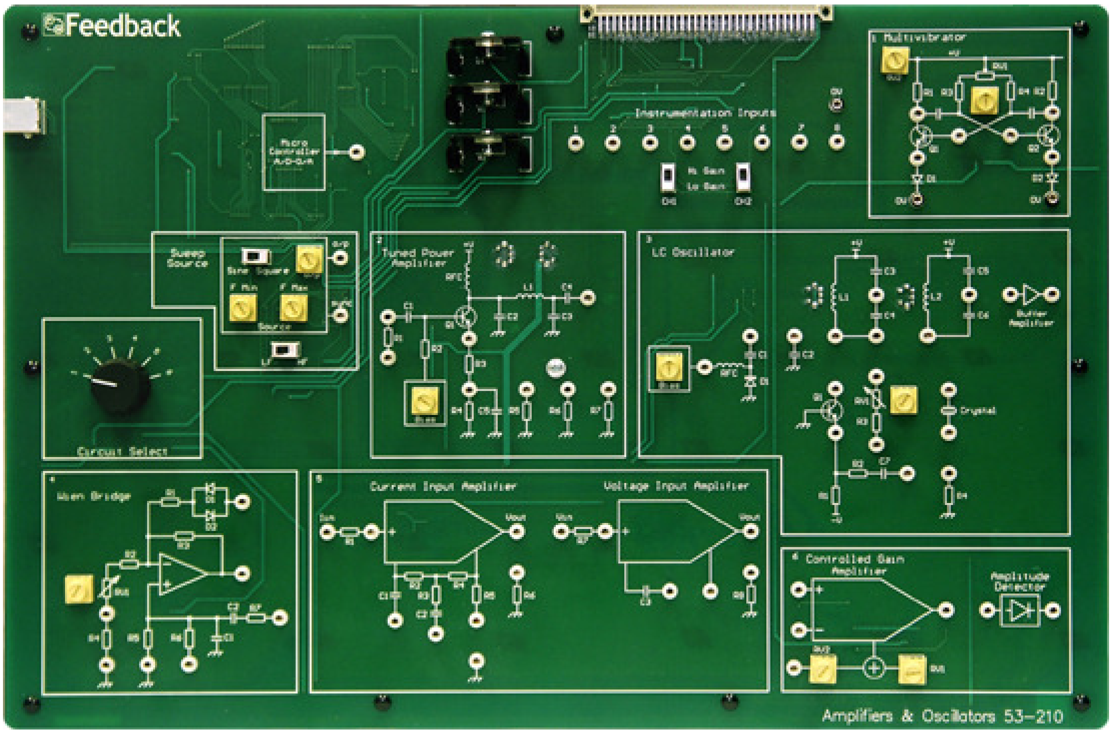
To become familiar with the interconnection of the workboard, terminal and PC

To determine that the set-up is functioning as required

To learn how to navigate the software

The Workboard – an Introduction

The Amplifiers & Oscillators 53-210 workboard contains a number of circuit blocks that may be interconnected in many ways to demonstrate the principles and operation of typical amplifier and oscillator circuits used in modern telecommunications equipment.



The workboard is designed to operate with the Real-Time Access Terminal (RAT) 92-200, into which it plugs to obtain power and to provide, in conjunction with a personal computer (PC), the instrumentation required by the assignments.

Both the workboard and the RAT require USB connection to the PC.

Interconnection between the various circuit blocks on the workboard is by 2mm, stackable patchleads. It is recommended that no more than two leads be stacked, as more than this is mechanically vulnerable and can lead to damage of the lead or the workboard.

Practical 1: The Circuits Available

Objectives and Background

This Practical is an exercise to get you conversant with the circuit blocks that are available on the Amplifiers & Oscillators workboard. There is no patching or measurement associated with this Practical.

At this stage, do not worry if you do not understand the description or function of the circuit blocks. As you progress through the assignments their functions and operation should become clearer.

Practical 1: The Circuits Available

Perform Practical

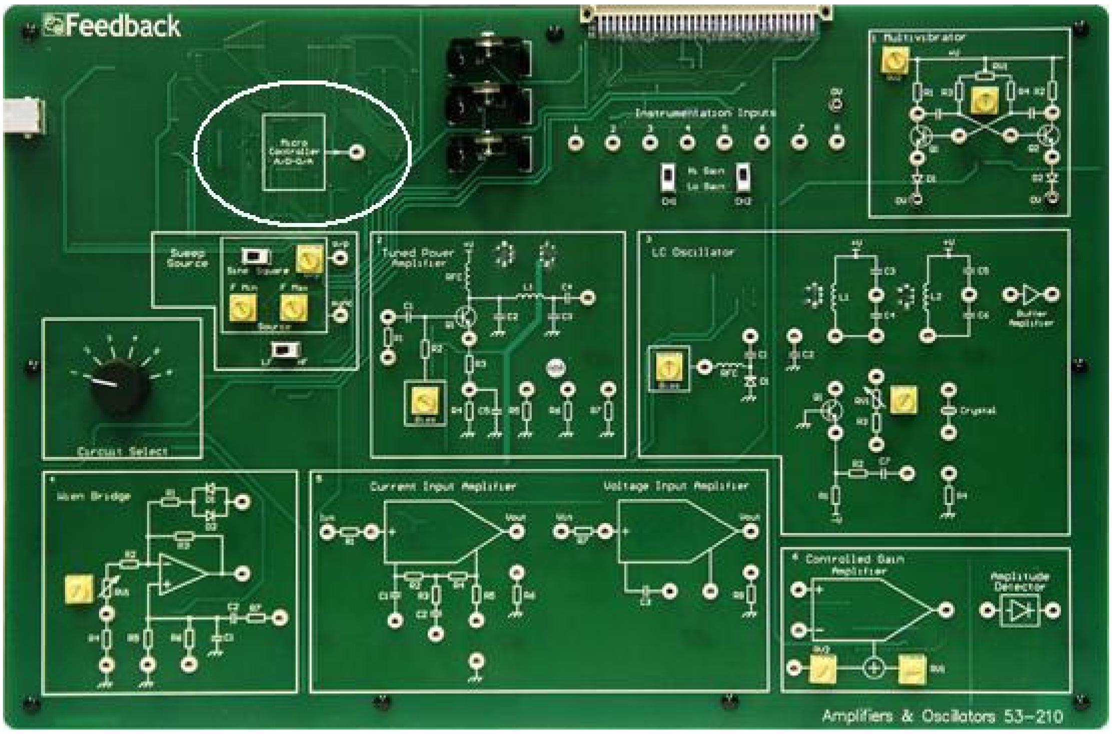
This Practical requires no workboard patching connections and there are no measurements to be taken.

Read through the descriptions below and identify each of the circuit blocks described.

At this stage, do not worry if you do not understand the description or function of the circuit blocks. As you progress through the assignments their functions and operation should become clearer.

The Micro Controller

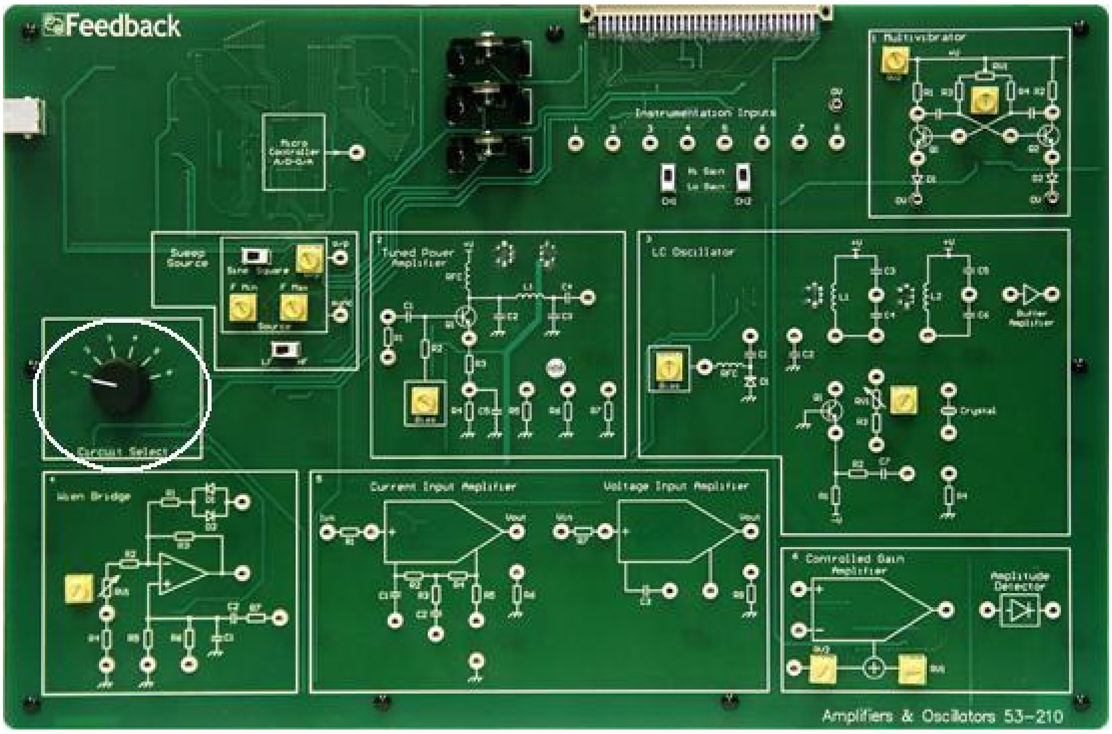
Towards the top left-hand corner of the workboard you will see the Micro Controller and A/D-D/A circuit block.



This block contains the circuitry and firmware that provides the modulation source for many of the assignments. It also provides waveforms and timing signals for a number of the assignments.

The Circuit Select Switch

To the middle left-hand side of the workboard you will see the Circuit Select switch. It is a 6-way rotary switch and its function is to select the circuit block to which dc power is applied.



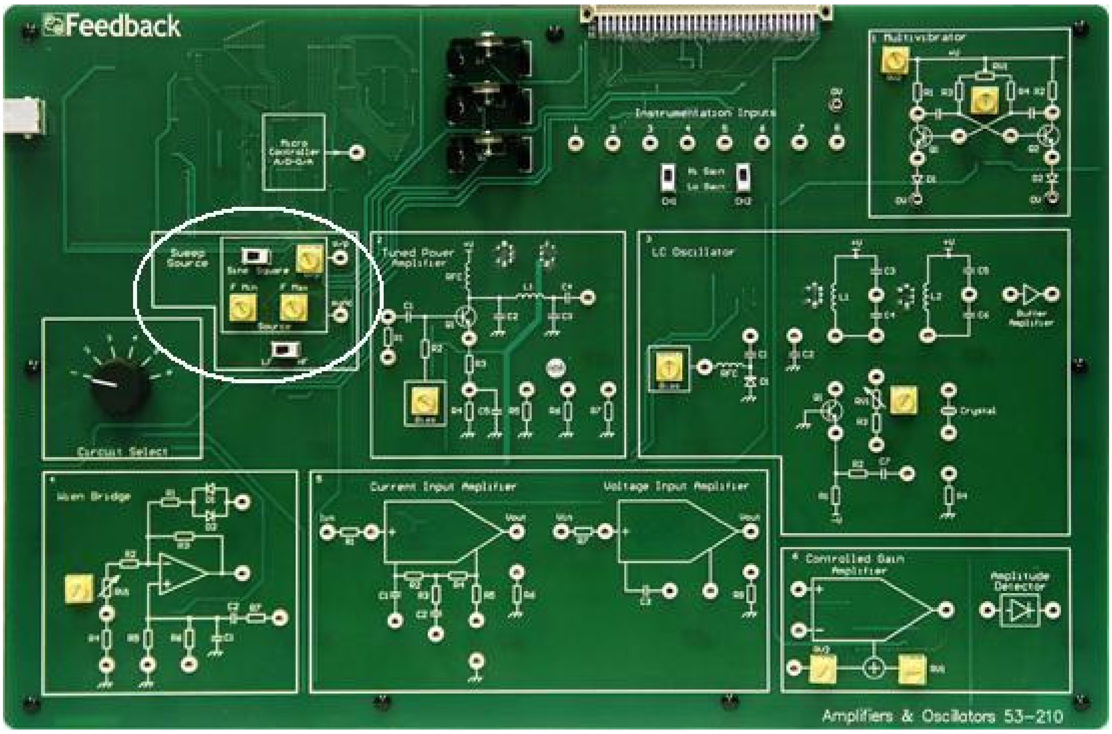
The reason for having this switch is to ensure that unused circuit blocks are not powered up and do not produce unwanted interfering signals that might affect the results you achieve from the circuit that you are investigating.

The positions on the switch are labelled 1 to 6 and correspond to the numbers found in the top left-hand corner of each of the circuit blocks on the workboard. The micro Controller and Sweep Source circuit blocks are required in many of the assignments and are continuously powered.

The Sweep Source

To the right of the Circuit Select switch you will find the Sweep Source circuit block.

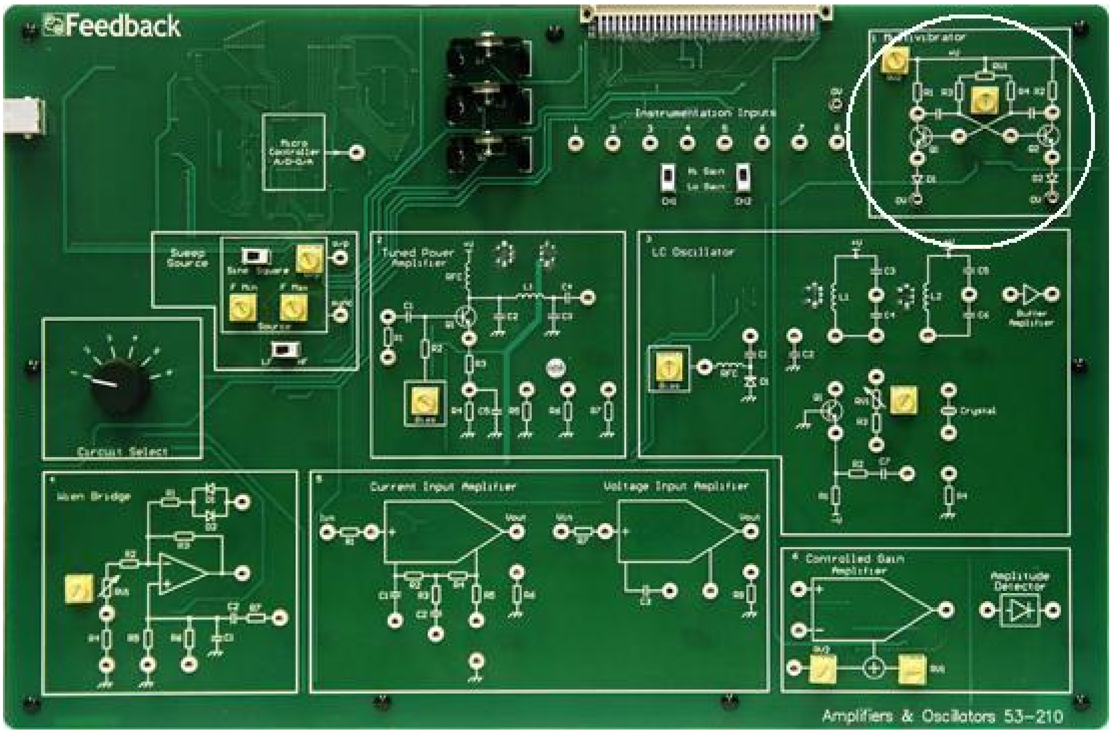
This is a signal source that may be swept in frequency from approximately 300Hz to 8MHz, in two ranges, selectable by slide switch. The range of frequency over which the source sweeps is set by the **FMin** and **FMax** controls.



The source can produce either sine or square wave outputs, selectable by slide switch, and the output amplitude is variable using the **o/p** control.

The Multivibrator

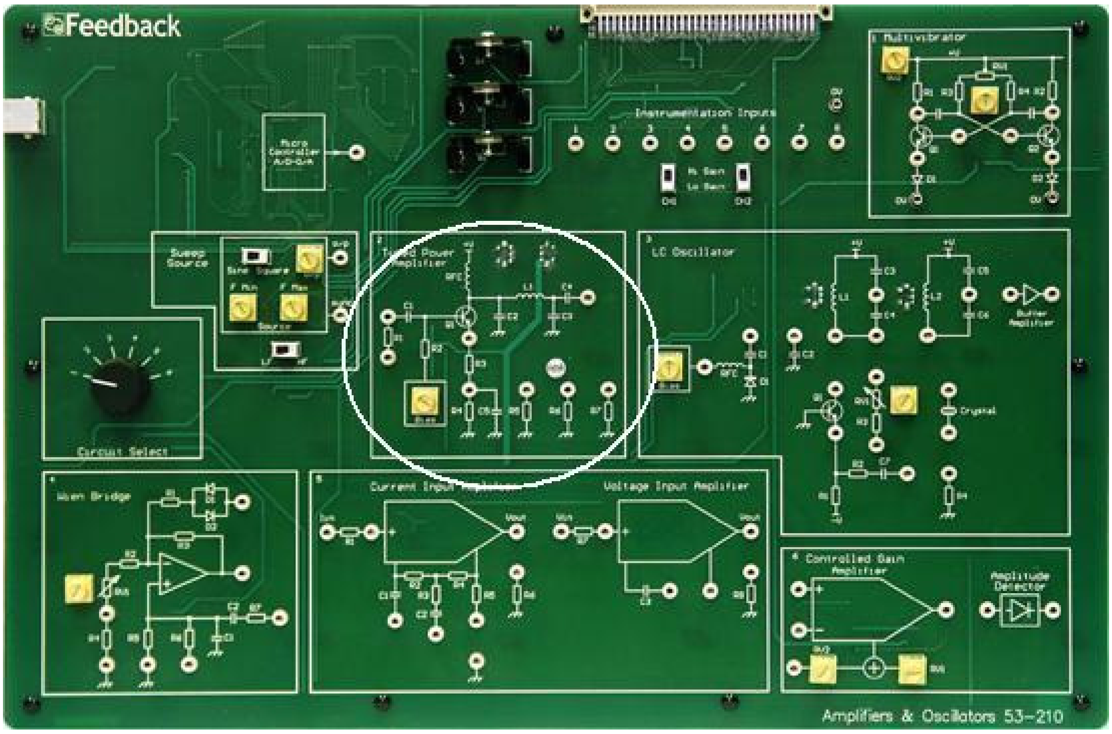
This circuit block is positioned in the top right-hand corner of the workboard.



A multivibrator is a form of oscillator circuit often used in digital circuits to produce a square waveform output.

The Tuned Power Amplifier

This circuit block is to be found in the centre of the workboard.

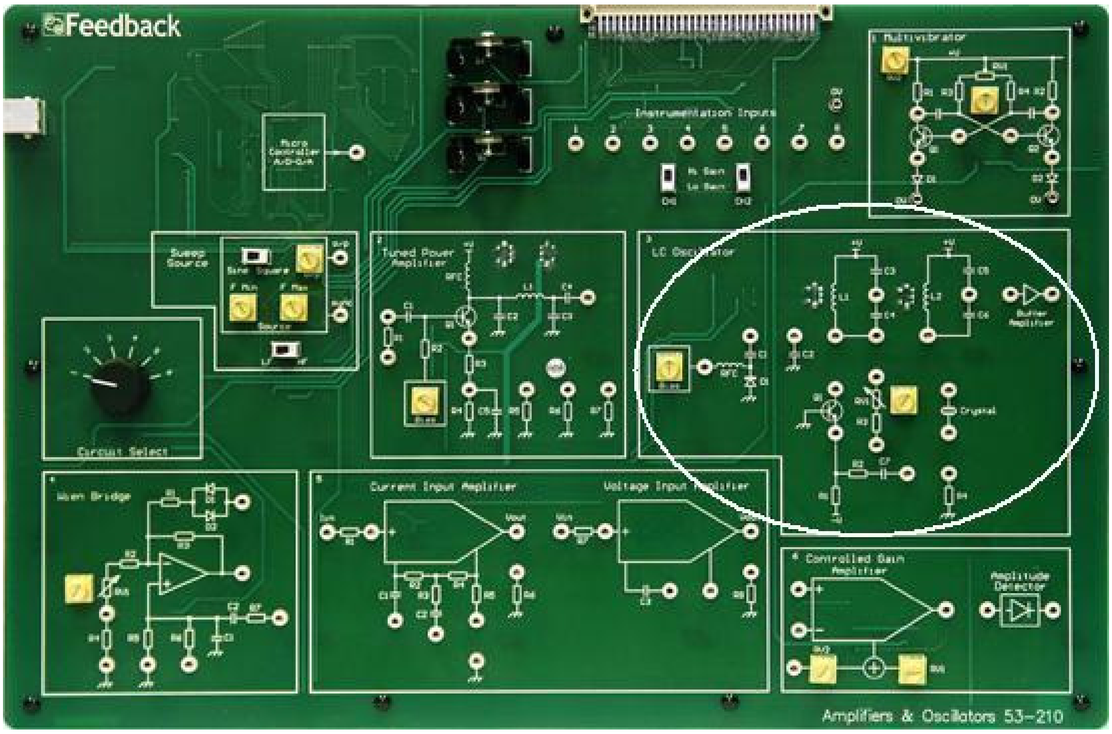


To amplify the power of a signal is a common requirement in electronics and telecommunications. Many communications applications operate at high frequencies and require amplifiers that are frequency selective. These are often produced using resonant (or tuned) circuits to provide the selectivity.

The circuit in this block is used to investigate the principles and operation of these tuned power amplifiers.

The LC Oscillator

This circuit block is positioned to the right-hand centre of the workboard.

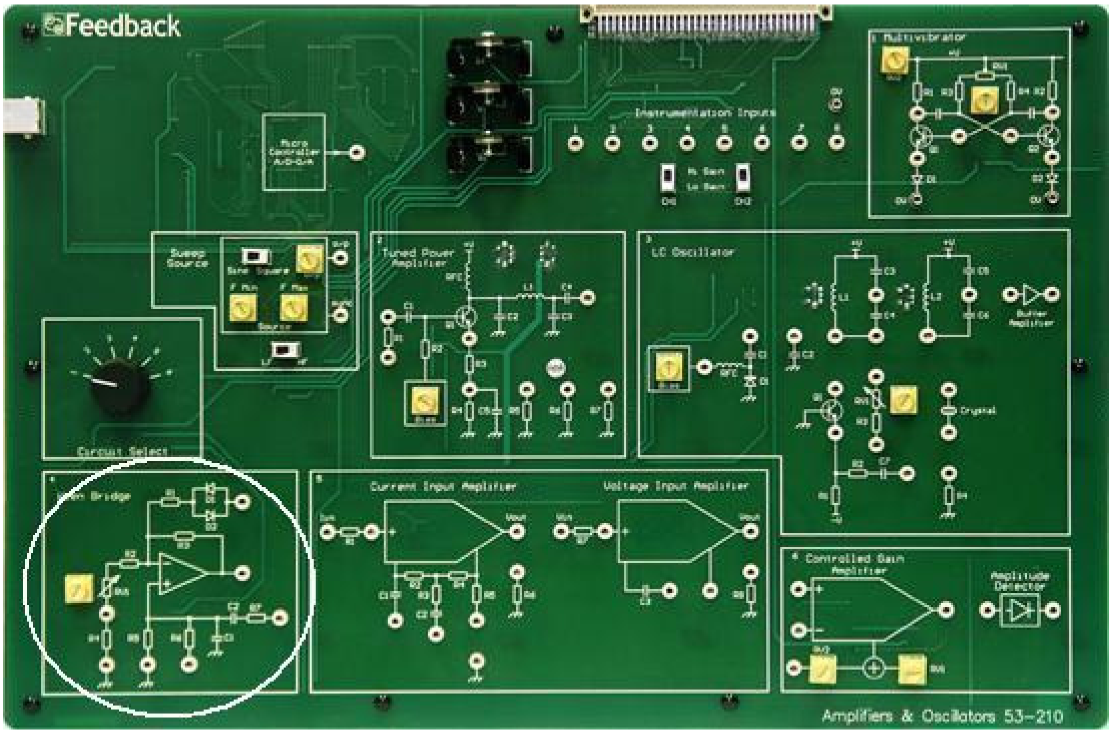


The LC (inductance/capacitance) form of oscillator circuit is a commonly used signal source circuit found in high frequency telecommunications applications.

You will investigate the general requirements for oscillation in electronic oscillators and how these are achieved in this circuit.

The Wien Bridge

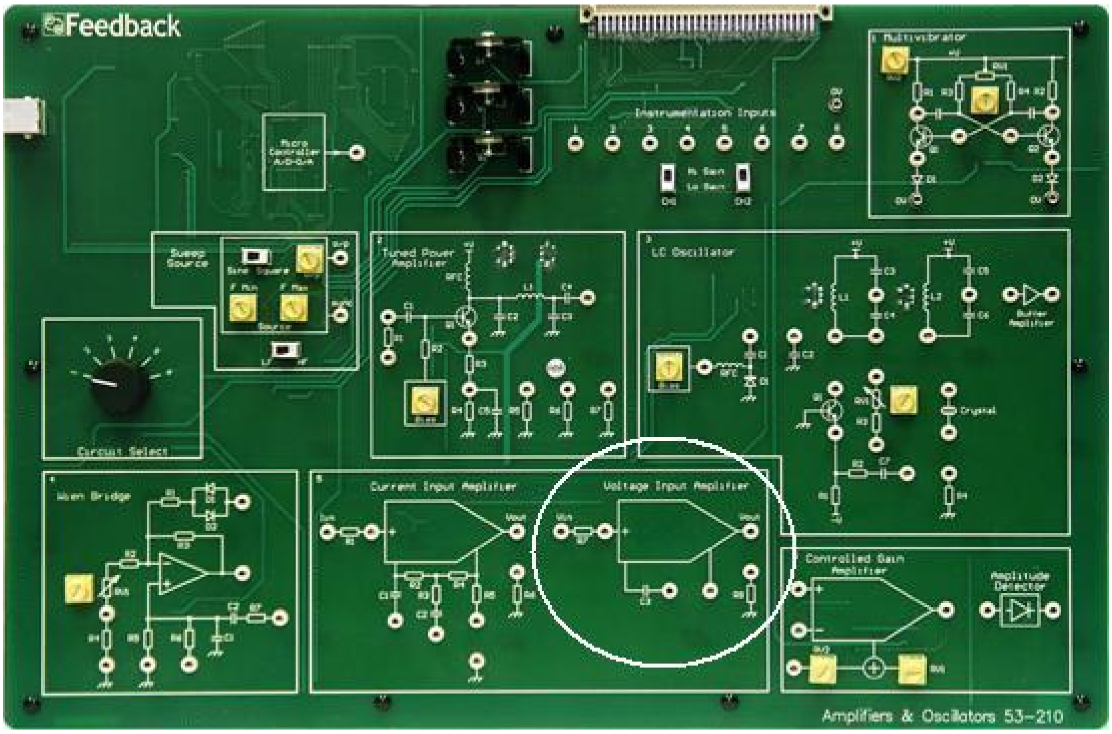
This circuit block is in the bottom left-hand corner of the workboard.



The Wien Bridge circuit is another example of an oscillator, or signal source, circuit. It is most often used at relatively low frequencies (c.f. the LC oscillator). The circuit on the workboard can operate between approximately 300Hz and 8MHz, determined by the setting of the control RV_1.

The Voltage Input Amplifier

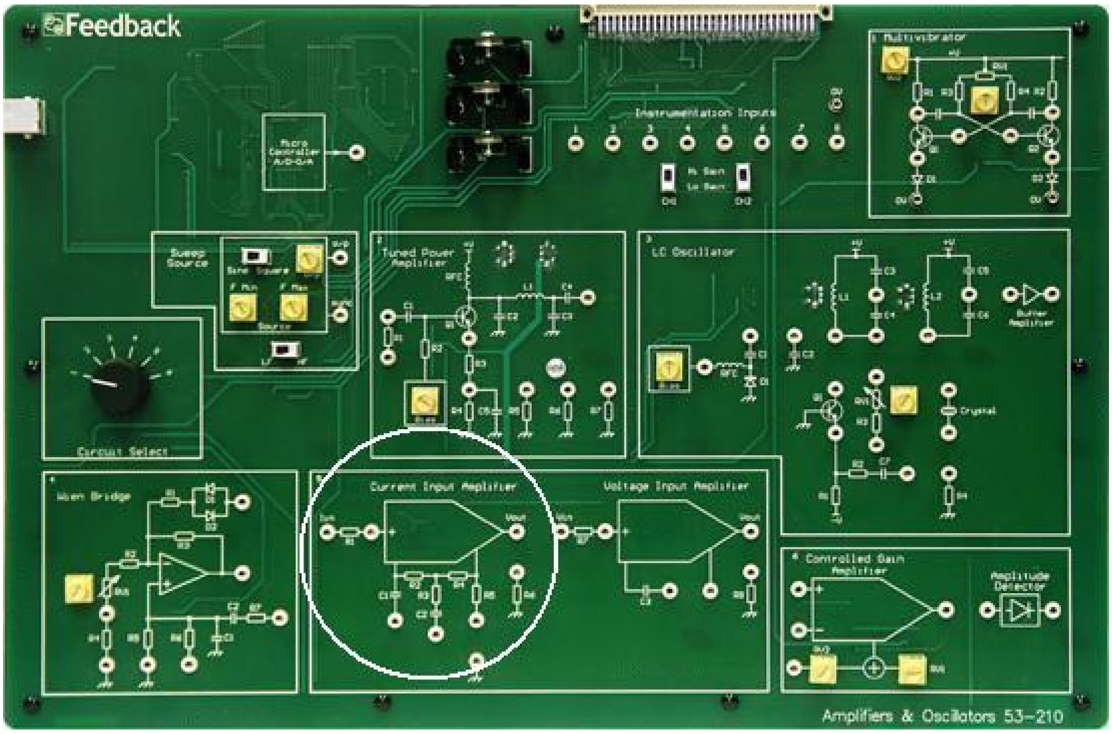
This is the right-hand circuit found in the lower central block on the workboard.



This circuit takes an input voltage signal and produces an amplified output voltage. Its requirements and performance will be studied.

The Current Input Amplifier

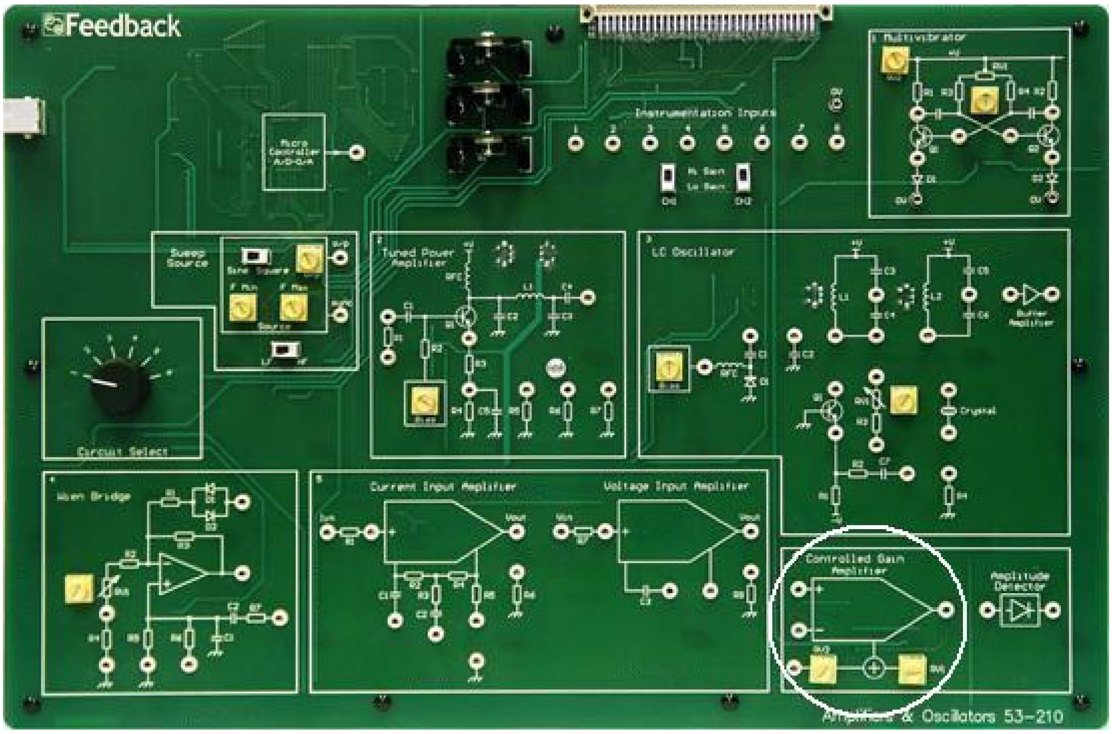
This is the left-hand circuit found in the lower central block on the workboard.



This circuit takes an input current signal and produces an amplified output voltage. It is more properly called a transresistance amplifier.

The Controlled Gain Amplifier

This is to be found in the bottom right-hand corner of the workboard.

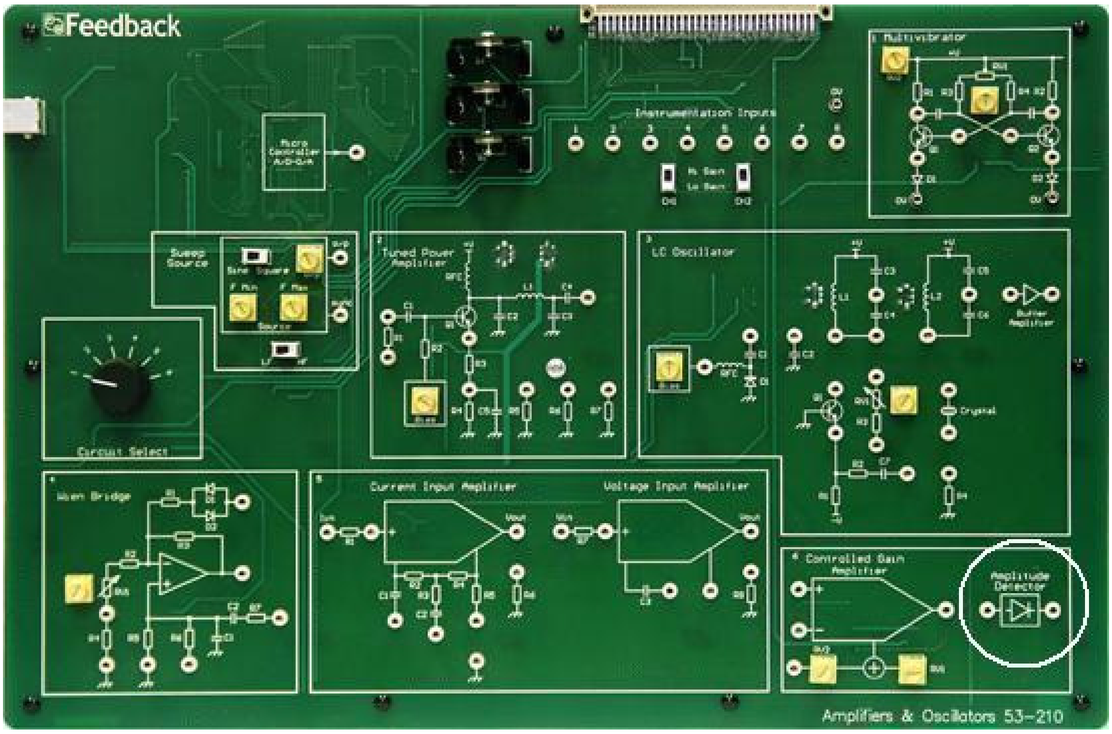


A Controlled Gain Amplifier is one in which its gain can be controlled by an applied control voltage. It is often called a gain controlled amplifier.

A common telecommunications application for such a circuit is as an AGC (automatic gain control) circuit used in radio receivers.

Amplitude Detector

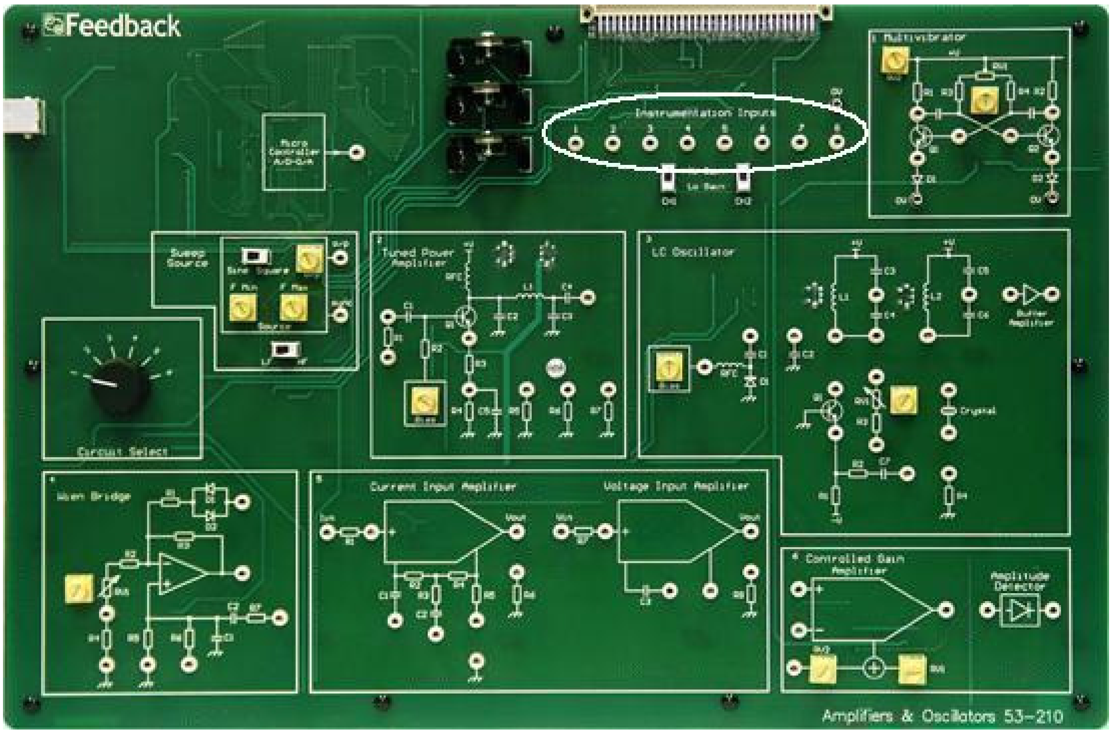
This circuit, found within the Controlled Gain Amplifier circuit block, is used with the associated amplifier to investigate an AGC circuit.



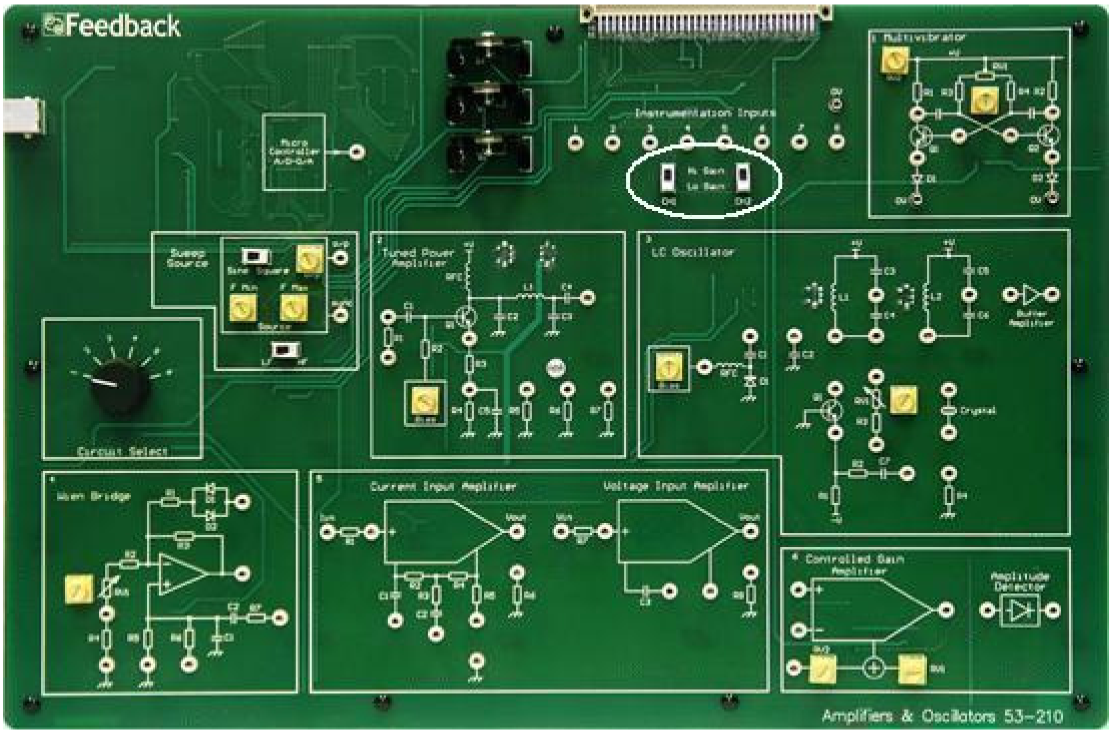
Instrumentation Inputs

Signals present at any of the sockets available on the workboard may be measured and displayed on a PC using a Real-Time Access Terminal (RAT) and the Discovery software that accompanies the product.

The points to be monitored must be patched to the Instrumentation Input sockets that are to be found at the top centre of the workboard. The figures associated with these sockets correspond to the numbers on the monitoring points as seen on the diagrams associated with each Practical activity.



Associated with the Instrumentation Inputs are two switches that switch the gain of the two instrumentation channels.



Practical 2: Connections to the PC

Objectives and Background

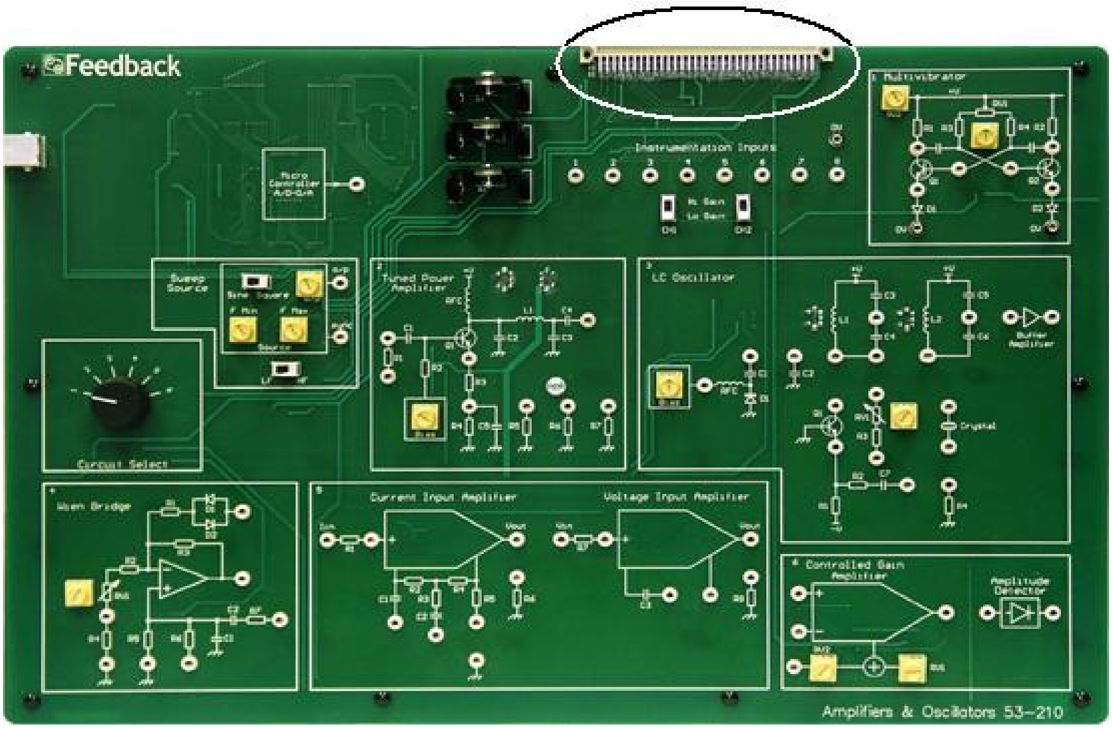
This Practical will familiarise you with the connections required to operate the Amplifiers & Oscillators 53-210 workboard with a PC.

Practical 2: Connections to the PC

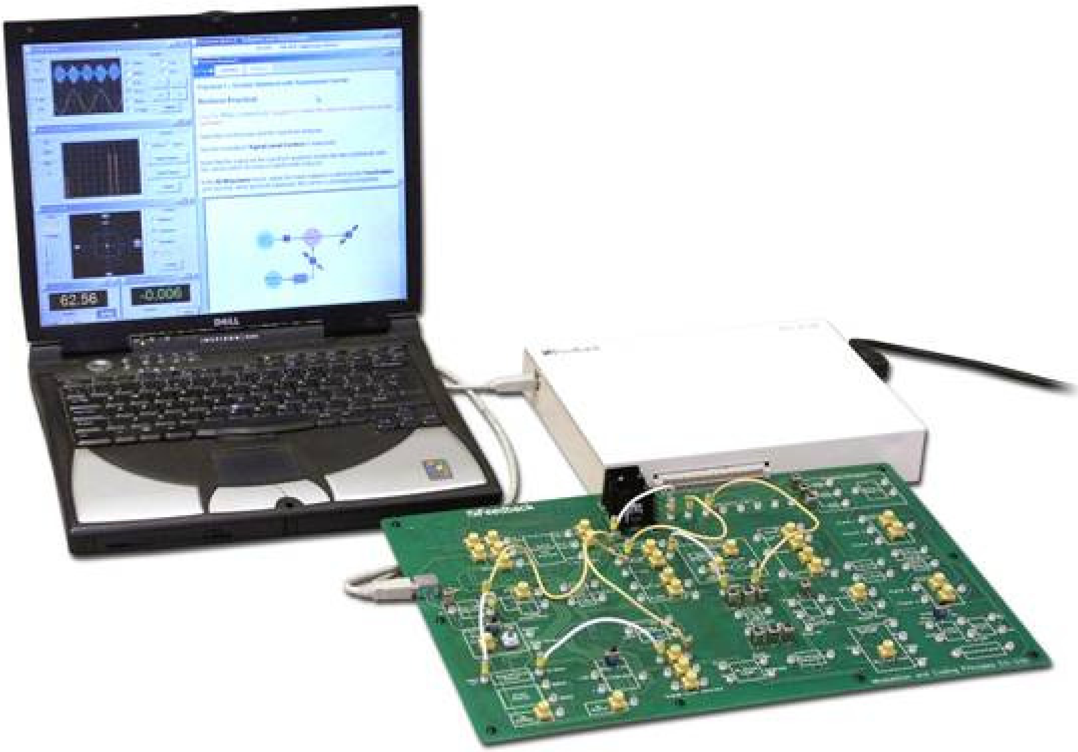
Perform Practical

This Practical requires no workboard patching connections and there are no measurements to be taken.

Identify the multiway connector on the top edge of the workboard.



This connector plugs into its female counterpart on the front edge of the Real-time Access Terminal (RAT) 92-200. The diagram below shows a workboard plugged into a RAT, together with a laptop PC.



Both the RAT and the workboard require USB connection to the PC. They may be USB1 or USB2 ports. If you do not have two available USB sockets on your PC, an external hub will have to be used. It may be either powered or un-powered.

For correct operation the PC must have the relevant Discovery software and the RAT and product drivers installed. If it does not, you will need to consult your tutor.

If the Discovery software has been installed the workboard and the RAT should automatically be recognised on switch-on and the system will be ready for use.

Practical 3: Operational Check

Objectives and Background

In this Practical you will perform a very simple operational check to confirm that the PC, the RAT and the workboard are communicating with each other and that the set-up is ready to perform a practical Assignment.

Practical 3: Operational Check

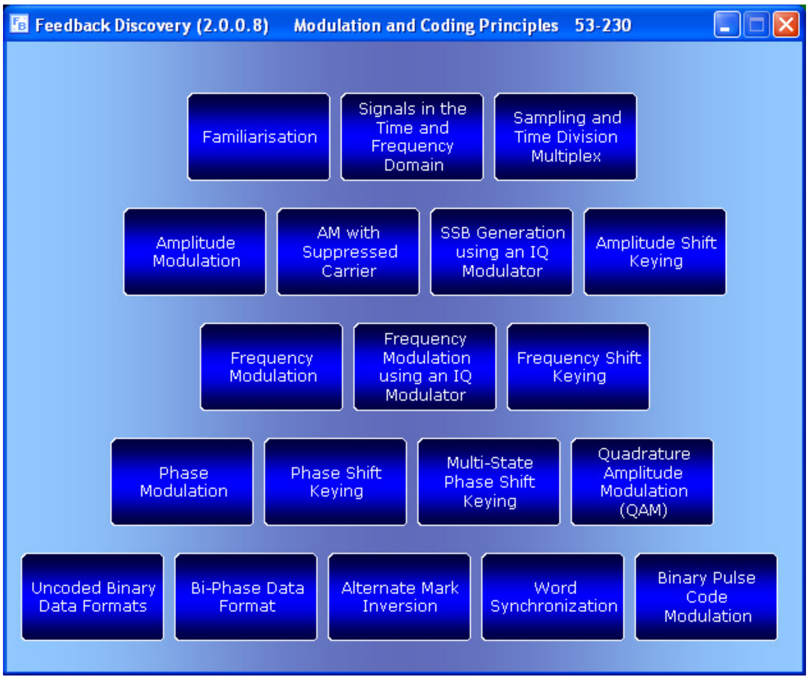
Perform Practical

This Practical requires no workboard patching connections and there are no measurements to be taken.

Ensure that you have connected the equipment as described in Practical 2 of this Assignment.

Ensure that the PC and the RAT are switched on.

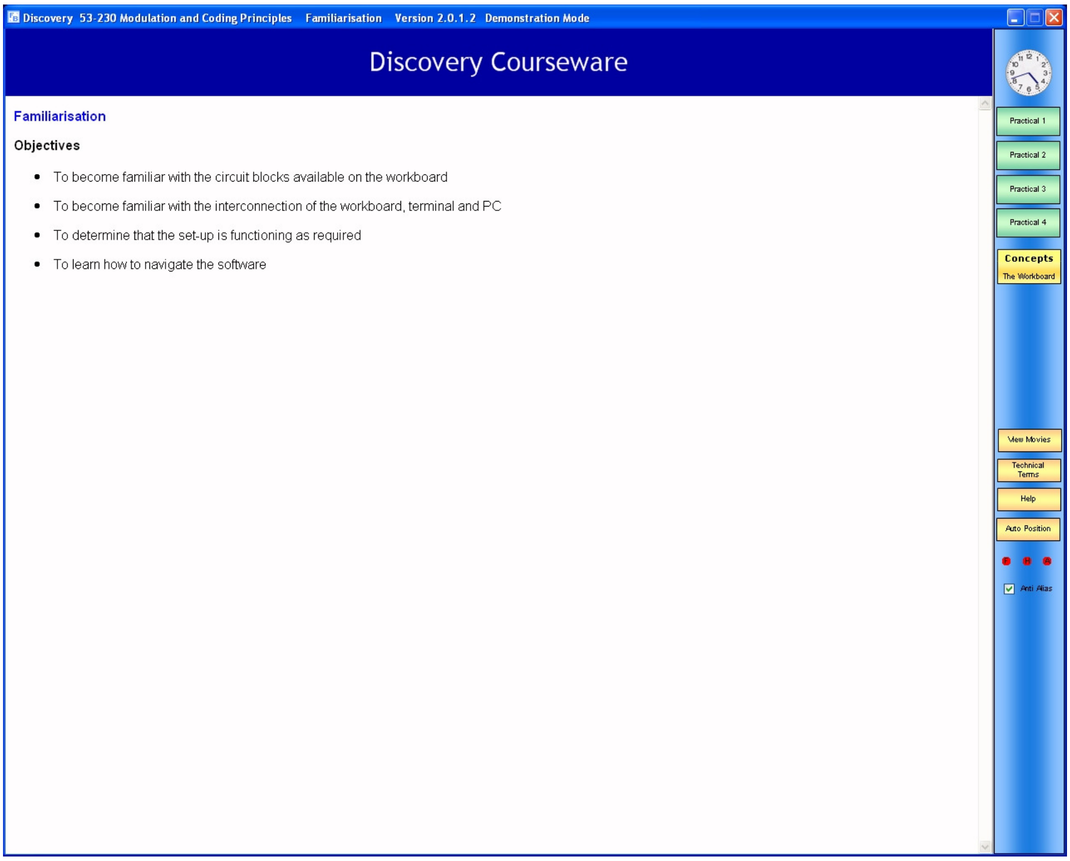
Launch the Discovery software associated with the product.



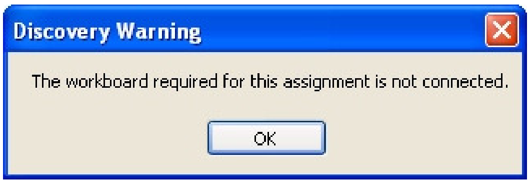
After a Discovery Courseware splash screen has been briefly displayed, you should see a window showing all the assignments that are available for the product, of the form shown above. There may be a smaller or greater number of assignments available to you than shown. The precise appearance of this window, such as the choice of colours and how the buttons are arranged, is determined by your tutor. Note that you cannot close this window whilst any assignment is open, and you can have only one assignment open at any time.

To select an assignment to perform, left-click on the appropriate button.

After an Assignment loading dialog has been briefly displayed, the assignment window should appear. The assignment window is full-screen, consisting of a title bar across the top, a side bar at the right-hand edge, and the main working area. Initially the overall objectives for the chosen assignment are shown in the main working area. A typical screen shot is shown below. The precise appearance of the assignment window is determined by your tutor.



If the hardware has not been connected properly, the following Discovery Warning message is immediately displayed on the screen:



If this warning message is shown, you must acknowledge it by clicking the OK button before you can continue. In this event, it is recommended that you resolve the problem before attempting to perform the assignment. You will need to close the assignment, correct the hardware problem and then restart the assignment.

On the screen shot of the assignment window, notice the three red indicators within the side bar. These are marked ‘F’, ‘H’ and ‘A’. These are warning indicators. If any one of them is visible on your screen then you have a fault condition, as follows:

F indicates that there is a firmware communications error;

H indicates that the hardware is incorrectly connected, probably your workboard is incorrectly connected to your PC, or that the workboard driver is not installed correctly;

A indicates that there is a data acquisition error, probably your RAT is incorrectly connected to the PC, or that the RAT driver is incorrectly installed.

If you do not see any of these warning indicators on your screen then your set-up is correct and you may perform any of the Practicals in the assignment. You can still open a Practical when a fault condition exists, but you will not be able to use any test equipment that may be required to perform that Practical. The hardware must be correctly connected before starting an assignment in order to use the test equipment in any of the Practicals within that assignment.

The next Practical takes you through the navigation of the software.

Practical 4: Navigating the Discovery Software

Objectives and Background

Although the Discovery Laboratory environment is very easy to operate, these notes will help you use all its facilities more quickly.

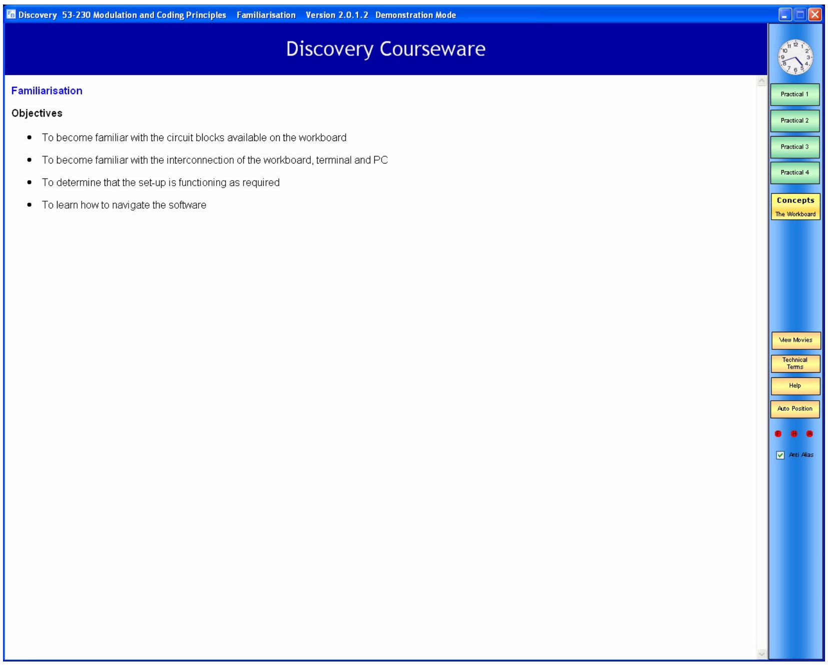
If there is a demonstration assignment, slider controls in the software perform functions that would normally be performed on the hardware. In normal assignments, if the any of hardware systems fail to initialise the system reverts to demonstration mode. This means that none of the test equipment is available.

Practical 4: Navigating the Discovery Software

Perform Practical

This Practical requires no workboard patching connections and there are no measurements to be taken.

The assignment window opens when an assignment is launched as described in the previous Practical. The assignment window consists of a title bar across the top, an assignment side bar at the right-hand edge, and the main working area. By default, the overall assignment objectives are initially shown in the main working area whenever an assignment is opened. The assignment window occupies the entire screen space and it cannot be resized (but it can be moved by ‘dragging’ the title bar, and it can be minimised to the task bar). The title bar includes the name of the selected assignment. The side bar contains the Practicals and any additional resources that are relevant for the selected assignment. The side bar cannot be repositioned from the right-hand edge of the assignment window. An example of an assignment window is shown below.



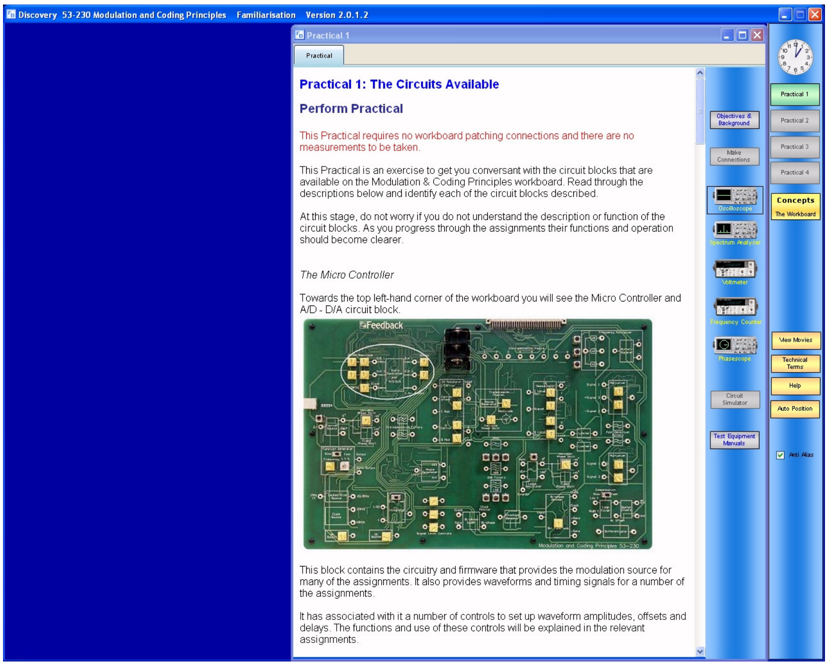
The precise appearance of the assignment window will depend on the ‘skin’ that has been selected by your tutor. However, the behaviour of each of the buttons and icons will remain the same, irrespective of this.

The clock (if you have one active) at the top of the side bar retrieves its time from the computer system clock. By double clicking on the clock turns it into a stop watch. To start the stop watch single click on the clock, click again to stop the stop watch. Double clicking again will return it to the clock function.

There are a number of resource buttons available in the assignment side bar. These are relevant to the selected assignment. In general, the resources available will vary with the assignment. For example, some assignments have video clips and some do not. However, the Technical Terms, Help and Auto Position buttons have identical functionality in every assignment. You can click on any resource in any order, close them again, or minimise them to suit the way you work.

Practicals are listed in numerical order in the side bar. When you hover the mouse over a Practical button, its proper title will briefly be shown in a pop-up tool-tip. There can be up to four Practicals in any assignment. You can have only one Practical window open at any time.

To perform a Practical, left-click on its button in the assignment side bar. The assignment objectives, if shown in the main working area, will close, and the selected Practical will appear in its own window initially on the right-hand side of the main working area, as shown below. You can move and resize the Practical window as desired (even beyond the assignment window) but its default size and position allows the test equipment to be displayed down the left-hand side of the main working area without overlapping the instructions for the Practical.



Again, the precise appearance of the Practical window can be determined by your tutor but the behaviour of each of the buttons and icons will remain the same, irrespective of this. Whatever it looks like, the Practical window should have icons for the test equipment, together with buttons for Objectives & Background, Make Connections, Circuit Simulator and Test Equipment Manuals. These resources are found in side bar, located on the right-hand edge of the Practical window. The resources will depend on which Practical you have selected. Therefore not all the resources are available in every Practical. If a resource is unavailable, it will be shown greyed out. To open any resource, left-click on its icon or button. Note that when you close a Practical window, any resources that you have opened will close. You may open any resource at any time, provided it available during the Practical. The Circuit Simulator will only be available if you have one loaded.

Note that if the hardware is switched off, unavailable, or its software driver is not installed, all the test equipment is disabled. However, you can open any other window. If you switch on the hardware it will be necessary to close the assignment window and open it again to enable the test equipment.

Resource Windows

These are windows may be moved, resized and scrolled. You may minimise or maximise them. The system defaults to 'Auto Position', which means that as you open each resource window it places it in a convenient position. Most resource windows initially place themselves inside the practical window, selectable using tabs. Each one lays over the previous one. You can select which one is on top by clicking the tab at the top of the practical window. You can see how many windows you have open from the number of tabs. If you want to see several windows at once then drag them out of the practical window to where you wish on the screen. If you close a window it disappears from the resources tab bar.

If you want to return all the windows to their default size and position simply click the Auto Position button in the assignment side bar.

Make Connections Window

This movable and resizable window shows the wire connections (2mm patch leads) you need to make on the hardware to make a practical work. Note that some of the wires connect the monitoring points into the data acquisition switch matrix. If this is not done correctly the monitoring points on the practical diagram will not correspond with those on the hardware. The window opens with no connections shown. You can show the connections one by one by clicking the Show Next button or simply pressing the space bar on the keyboard. If you want to remove the connections and start again click the Start Again button. The Show Function button toggles the appearance of the block circuit diagram associated with the Practical.

Test Equipment

The test equipment will auto-place itself on the left of the screen at a default size. You may move it or resize it at any time. Note that below a useable size only the screen of the instrument will be shown, without the adjustment buttons. Each piece of test equipment will launch with default settings. You may change these settings at any time. There is an auto anti-alias feature that prevents you setting time-base or frequency settings that may give misleading displays. If auto anti-alias has operated the button turns red. You can turn off the anti-aliasing feature, but you should be aware that it may result in misleading displays.

You may return to the default settings by pressing the Default button on each piece of test equipment. If you wish to return all the equipment to their original positions on the left of the screen click Auto Position on the side bar of the assignment window.

Note that if you close a piece of test equipment and open it again it returns to its default position and settings.

If you want more information on how a piece of test equipment works and how to interpret the displays, see the Test Equipment Manuals resource in the Practical side bar.

On slower computers it may be noticeable that the refresh rate of each instrument is reduced if all the instruments are open at once. If this is an issue then only have open the instrument(s) you actually need to use.

Test Equipment Cursors

If you left click on the display of a piece of test equipment that has a screen, a green cursor marker will appear where you have clicked. Click to move the cursor to the part of the trace that you wish to measure. If you then move the mouse into the cursor a tool-tip will appear displaying the values representing that position. Note if you resize or change settings any current cursor will be removed.

Perform Practical Window

This window contains the instructions for performing the practical, as well as a block, or circuit, diagram showing the circuit parts of the hardware board involved in the Practical. On the diagram are the monitoring points that you use to explore how the system works and to make measurements. The horizontal divider bar between the instructions and the diagram can be moved up and down if you want the relative size of the practical instruction window to diagram to be different. Note that the aspect ratio of the diagram is fixed.

Information Buttons on Practical Diagrams

On many of the symbols on the diagram you will find a button that gives access to new windows that provide more information on the circuit that the symbol represents. Note that these windows are "modal", which means that you can have only one open at a time and you must close it before continuing with anything else.

A Further Information point looks like this 

Probes

The practical diagram has probes on it, which start in default positions. These determine where on the hardware the signals are being monitored.

Selecting and Moving the Probes

Probes are indicated by the coloured icons like this .

If this probe is the *selected probe* it then looks like this  (notice the black top to the probe). You select a probe by left clicking on it.

Monitor points look like this 

If you place the mouse over a monitor point a tool-tip will show a description of what signal it is.

You can move the selected probe by simply clicking on the required monitor point. If you want to move the probe again you do not have to re-select it. To change which probe is selected click on the probe you want to select.

You can also move a probe by the normal ‘drag-and-drop’ method, common to ‘Windows’ programs.

Probes and Test Equipment Traces

The association between probes and traces displayed on the test equipment is by colour. Data from the blue probe is displayed as a blue trace. Yellow, orange and green probes and traces operate in a similar way. Which piece of test equipment is allocated to which probe is defined by the practical.

Note that the phasescope shows the relative phase and magnitude of the signal on its input probe using another probe as the reference. The reference probe colour is indicated by the coloured square to the top left corner of the phasescope display.

Practical Buttons

On some Practicals there are buttons at the bottom of the diagram that select some parameter in the practical. These can be single buttons or in groups. Only one of each button in a group may be selected at one time.

Slider Controls

Where slider controls are used you may find you can get finer control by clicking on it and then using the up and down arrow keys on your keyboard.

The Voltage Amplifier

Objectives

To appreciate the concept of a voltage amplifier

To investigate the gain and the phase shift associated with a voltage amplifier

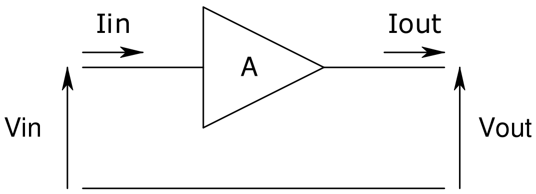
To determine the typical input resistance of a voltage amplifier

To determine the typical output resistance of a voltage amplifier

Amplifiers

Gain

The block diagram of a general amplifier is given below.



The gain of the amplifier is denoted by the symbol A. The definition of A is given by:

$$\text{Gain, A} = \frac{(output\ power)}{(input\ power)}$$

This is called the **power gain** of the amplifier.

If a circuit does not have power gain, then it is not an amplifier!

As you can see, there is present input current and input voltage and, at the output, the corresponding output current and voltage. As power is the product of current and voltage, this gives the expression for the gain:

$$A = \frac{Vout.Iout}{Vin.Iin}$$

Now:

$$\frac{Vout}{Vin} = A_V$$, the **voltage gain** of the amplifier

and:

$$\frac{Iout}{Iin} = A_I$$, the **current gain** of the amplifier

Therefore:

$$A = A_V \,.\, A_I$$

Note: that it is quite possible for an amplifier to have a voltage gain of less than one; however, its corresponding current gain must be high enough to give a power gain greater

than unity for it to be classed as an amplifier. This also works the other way: if the current gain is less than one, the voltage gain must be high enough.

Going back to the expression for gain of:

$$A = \frac{Vout.Iout}{Vin.Iin}$$

This can be further re-arranged by defining two more relationships:

$\frac{Vout}{Iin} = G_m$, the **transresistance gain** of the amplifier

and:

$\frac{Iout}{Vin} = R_m$, the **transconductance gain** of the amplifier

Giving, therefore:

$$A = G_m \, . \, R_m$$

Using a similar argument to before, to qualify as an amplifier, it is quite possible for either one or the other of these terms to be less than one providing the product of the two is greater than unity.

Types of Amplifier

The input signal to an amplifier may be a current or it may be a voltage. Therefore, this gives rise to two types of amplifier: the current input amplifier and the voltage input amplifier.

Each of these types of amplifier may be further sub-divided, as each can give a current output or a voltage output.

There are, therefore, four general forms of amplifier.

Different applications require amplifiers to have different properties of amplification.

An example

Consider a fibre optic communications system with a great distance between transmitter and receiver. Because of the attenuation of light along the length of fibre it is often necessary to compensate for this loss by having ‘repeater’ circuits at intervals along the fibre. The purpose of these circuits is to detect the incoming light signal, convert it into an electrical signal that can be amplified and then used to drive a secondary light source that

provides a regenerated signal for transmission further down the cable.

A typical sensor for the input of such a repeater is a photodiode. This will produce a current that is proportional to the light intensity. A typical output device might be a light-emitting diode, or perhaps a laser diode, which also needs a current signal to drive it. The amplifier within the repeater must thus take the small current output from the photodiode and amplify it to drive the LED or laser.

Therefore, the amplifier required has to be a **current amplifier**. The requirements for such a circuit are:

Its input should affect the signal current from the photodiode as little as possible,

It should have output circuitry that maximises the current transfer out of the amplifier,

It should have current gain.

To achieve this, it should have **as low an input resistance as possible** and its output should look like an ideal current source (i.e. it should have **as high an output resistance as possible**).

Another example

Consider a communications receiver system. Within the receiver there are circuits that perform functions such as high frequency amplification of the signals, the selection of the required signal (filters) and frequency translation of the signal to a lower frequency (mixers). Because of current limitations on the operating speed of analogue-to-digital converters and other digital circuitry, these functions are normally performed by analogue circuits. However, it is common practice to convert high frequency signals to a much lower frequency (often a few tens of kHz) so that such functions as demodulation or decoding and the final signal processing can be done using digital techniques. DSP (digital signal processing) circuits are widely used to do this.

Typically, DSP chips require input voltage signals of a few volts amplitude. The output signal from the analogue part of the receiver system may be only tens of millivolts in amplitude. An amplifier is therefore required to ‘bridge this gap’.

Therefore, this amplifier has to be a **voltage amplifier**. The requirements for such a circuit are:

Its input should affect the signal voltage from the analogue circuitry as little as possible,

It should have output circuitry that maximises the voltage transfer out of the amplifier,

It should have voltage gain.

To achieve this, it should have **as high an input resistance as possible** and its output should look like an ideal voltage source (i.e. it should have **as low an output resistance as possible**).

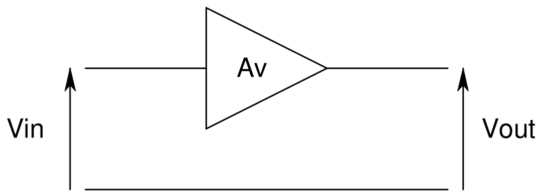
You can imagine that the requirements for the two example amplifiers above will result in completely different circuitry to satisfy them.

The two other types of amplifier also find uses in electronics and communications systems. For example, a field effect transistor amplifier is an example of a **transconductance amplifier** and **transresistance amplifiers** are widely used in audio

mixers and digital-to-analogue converters.

The Voltage Amplifier

A **voltage amplifier** is one to which an input voltage is applied and an output voltage results. The block diagram for such a system is given below.



The **voltage gain** of such an amplifier is given by

$$\text{Voltage gain,} \quad A_v = \frac{V_{out}}{V_{in}}$$

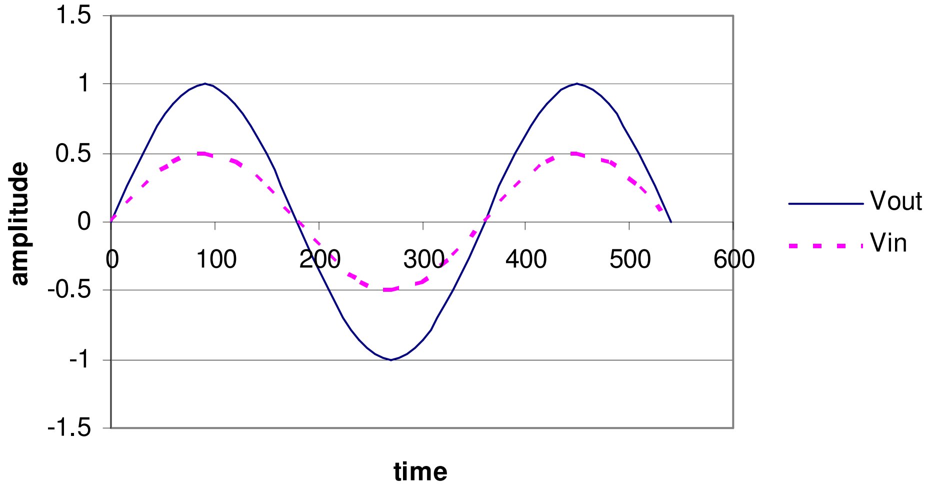
Ideally, connecting a voltage amplifier into a circuit should not affect the input voltage in any way. To achieve this, the input impedance of the amplifier should be as high as possible.

Also, ideally, connecting a load onto the output of the amplifier should not change the output voltage in any way. For this to happen, the output impedance of the amplifier should be as low as possible.

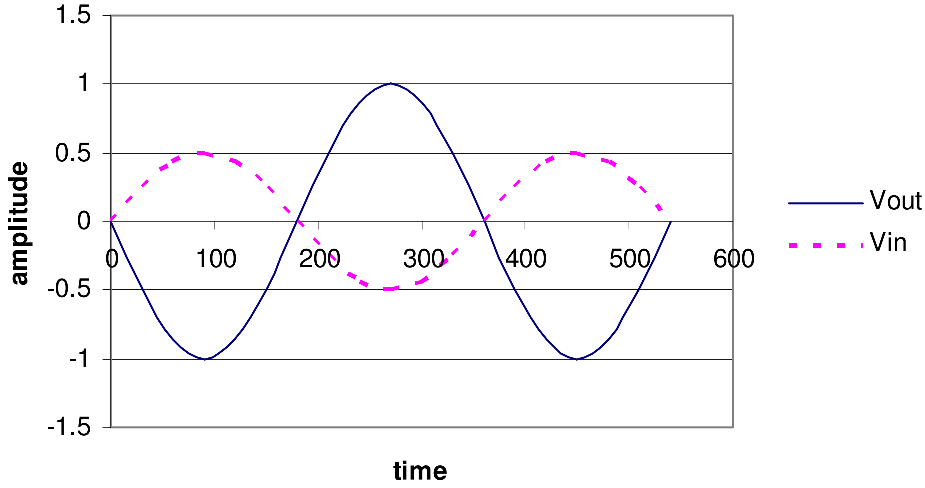
The ideal properties for a voltage amplifier are thus:

Voltage gain	Yes
Input impedance	Very high
Output impedance	Very low

The output voltage may follow the input voltage directly, or it may be inverted in polarity. This is illustrated below:



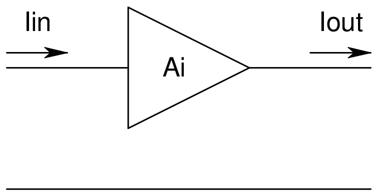
This shows the output following the input directly. The output is said to be **in phase** with the input. Another name for such an amplifier is a **non-inverting amplifier**. Note, also, that the output is amplified with respect to the input.



This shows the output inverted with respect to the input. The output is said to be in **anti-phase** with the input. Another name for such an amplifier is an **inverting amplifier**. Note, also, that the output is amplified with respect to the input.

The Current Amplifier

A **current amplifier** is one to which an input current is applied and an output current results. The block diagram for such a system is given below.



The **current gain** of such an amplifier is given by

$$\text{Current gain,} \quad A_i = \frac{I_{out}}{I_{in}}$$

Ideally, connecting a current amplifier into a circuit should not affect the input current in any way. To achieve this, the input impedance of the amplifier should be as low as possible.

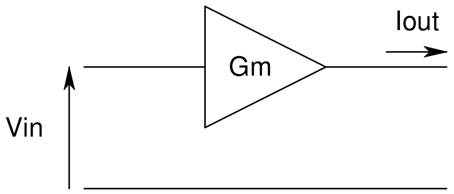
Also, ideally, connecting a load onto the output of the amplifier should not change the output current in any way. For this to happen, the output impedance of the amplifier should be as high as possible.

The ideal properties for a voltage amplifier are thus:

Current gain	Yes
Input impedance	Very low
Output impedance	Very high

The Transconductance Amplifier

A **transconductance amplifier** is one to which an input voltage is applied and an output current results. The block diagram for such a system is given below.



The **transconductance gain** of such an amplifier is given by

$$\text{Transconductance gain,} \quad G_m = \frac{I_{out}}{V_{in}}$$

Ideally, connecting a voltage amplifier into a circuit should not affect the input voltage in any way. To achieve this, the input impedance of the amplifier should be as high as possible.

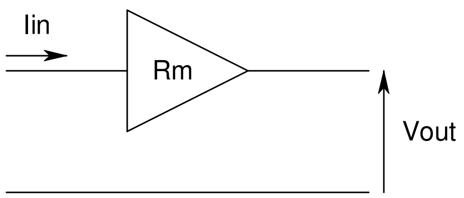
Also, ideally, connecting a load onto the output of the amplifier should not change the output current in any way. For this to happen, the output impedance of the amplifier should be as high as possible.

The ideal properties for a transconductance amplifier are thus:

Transconductance gain	Yes
Input impedance	Very high
Output impedance	Very high

The Transresistance Amplifier

A **transresistance amplifier** is one to which an input current is applied and an output voltage results. The block diagram for such a system is given below.



The **transresistance gain** of such an amplifier is given by

$$\text{Transresistance gain,} \quad R_m = \frac{V_{out}}{I_{in}}$$

Ideally, connecting a transresistance amplifier into a circuit should not affect the input current in any way. To achieve this, the input impedance of the amplifier should be as low as possible.

Also, ideally, connecting a load onto the output of the amplifier should not change the output voltage in any way. For this to happen, the output impedance of the amplifier should be as low as possible.

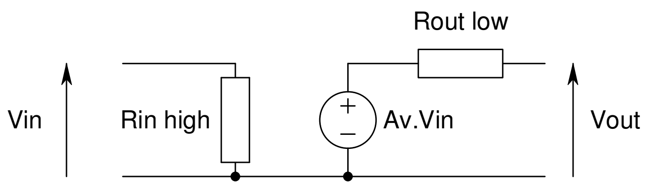
The ideal properties for a transresistance amplifier are thus:

Transresistance gain	Yes
Input impedance	Very low
Output impedance	Very low

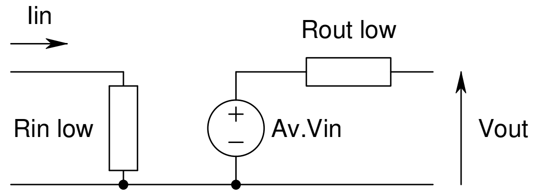
Summary of Amplifier Properties

A summary of the properties of the four types of amplifier, together with an equivalent circuit for each, is given below.

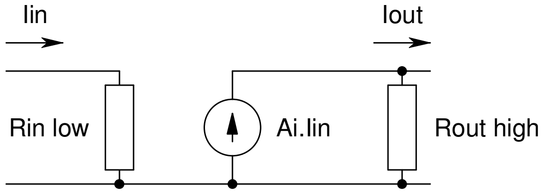
Amplifier Type	**Gain Required**	**Input Resistance**	**Output Resistance**
Voltage Amplifier	Voltage A_V	Very high	Very low
Current Amplifier	Current A_I	Very low	Very high
Transconductance Amplifier	Transconductance G_m	Very high	Very high
Transresistance Amplifier	Transresistance R_m	Very low	Very low



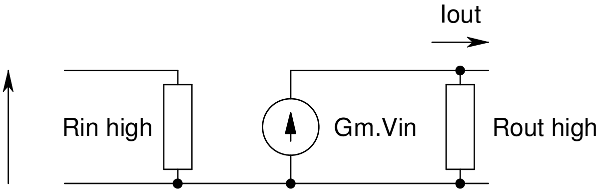
Voltage Amplifier



Transresistance Amplifier



Current Amplifier



Transconductance Amplifier

Gain, decibels and Input and Output Resistances

The following section uses some mathematics to explain how the decibel is used to describe the gain of an amplifier. Although it may look complex, working through it should make things much clearer. There is absolutely no need to commit these relationships to memory or be able to reproduce them. The important point is that you appreciate that this apparently complex concept is based on nothing more than Ohm's law and logarithms.

In the Introduction on Gain and phase, the power gain of a system in decibels was defined as:

$$10\log_{10}\frac{P_{out}}{P_{in}}\ \text{dB}$$

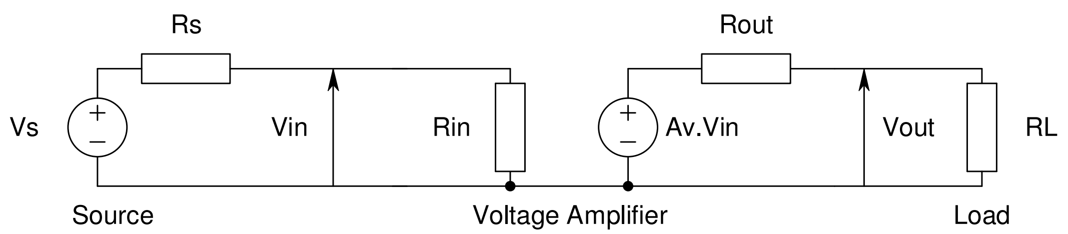
Expressing gain in dB should essentially be confined to the power gain, as the definition of the dB relates to power only. However, it was shown that the above expression became:

$$20\log_{10}\frac{V_{out}}{V_{in}}\ \text{dB} \qquad \textbf{if } \mathbf{R_L = R_{in}}.$$

So, in practice, the voltage gain of an amplifier is often expressed in dB, using this expression. Unfortunately, users often do not appreciate that normally R_L and R_{in} are normally **not** equal and that this inequality affects the gain.

By investigating the equivalent circuit of the amplifier, including its source and load, you can now see what happens if R_L and R_{in} are not equal.

The full equivalent circuit of a voltage amplifier, together with source and load, is given below.



From this:

$$P_{in} = \frac{V_{in}^2}{R_{in}} \text{ and } P_{out} = \frac{V_{out}^2}{R_L}$$

Therefore:

$$\text{Power gain} = \frac{P_{out}}{P_{in}} = \frac{V_{out}^2}{V_{in}^2}\cdot\frac{R_{in}}{R_L}$$

Expressing this is dB gives:

$$\text{Power gain} = 10\log_{10}\frac{P_{out}}{P_{in}} = 10\log_{10}\frac{V_{out}^2}{V_{in}^2}\cdot\frac{R_{in}}{R_L} \quad \text{dB}$$

$$= 20\log_{10}\frac{V_{out}}{V_{in}} + 10\log_{10}\frac{R_{in}}{R_L} \quad \text{dB}$$

From this expression you can see that, for a voltage amplifier, if the input and load resistances are **not** equal, the power gain in dB is **no longer** just 20 times the log of the voltage gain. There is the second term that is dependent on the ratio of those resistances.

If $\frac{R_{in}}{R_L} = 1$, then $10\log_{10}\frac{R_{in}}{R_L}$ will be zero and the power gain in dB will be the same as the voltage gain in dB.

If $\frac{R_{in}}{R_L} < 1$, then $10\log_{10}\frac{R_{in}}{R_L}$ will be negative and the power gain in dB will be less than the voltage gain in dB.

If $\frac{R_{in}}{R_L} > 1$, then $10\log_{10}\frac{R_{in}}{R_L}$ will be positive and the power gain in dB will be higher than the voltage gain in dB.

This shows that the gain of an amplifier depends on its input and load resistances. This is an important point to understand.

A similar analysis can be done for the other types of amplifier. The equations that result are given below.

Current amplifier:

$$\text{Power gain} = 20\log_{10}\frac{I_{out}}{I_{in}} + 10\log_{10}\frac{R_L}{R_{in}} \quad \text{dB}$$

$$\text{i.e. Power gain} = 20\log_{10} A_i + 10\log_{10}\frac{R_L}{R_{in}} \quad \text{dB}$$

Transconductance amplifier:

$$\text{Power gain} = 20\log_{10}\frac{I_{out}}{V_{in}} + 10\log_{10} R_L.R_{in} \quad \text{dB}$$

$$\text{i.e. Power gain} = 20\log_{10} G_m + 10\log_{10} R_L.R_{in} \quad \text{dB}$$

Transresistance amplifier:

$$\text{Power gain} = 20\log_{10}\frac{V_{out}}{I_{in}} + 10\log_{10}\frac{1}{R_L.R_{in}} \quad \text{dB}$$

$$\text{i.e. Power gain} = 20\log_{10} R_m + 10\log_{10}\frac{1}{R_L.R_{in}} \quad \text{dB}$$

These also show that the gain of an amplifier depends on its input and load resistances.

Thus the dB value of the power gain and that of the voltage gain (or current, transconductance or transresistance gain, depending on the type of amplifier) will only be equal when the input and load resistances are equal.

As you can now see, stating the voltage or current gain of an amplifier in decibels without some knowledge of the input resistance and the output load can well be misleading.

However, the use of $20\log_{10} \frac{V_{out}}{V_{in}}$ · has been adopted in practice as a convenient definition of the voltage gain of an amplifier (in dB), regardless of the magnitudes of the input and output resistances. This is a case of engineering "slang", and worse, is often done by those who do not appreciate how misleading it can be.

Gain and Phase

Gain

If a signal is applied to an electronic system, the output of that system is unlikely to be exactly the same as the input. For instance, the output is likely to be different in magnitude.

For example, the signal input voltage at the antenna of a radio receiver may be ten microvolts (10μV), whereas the output voltage that drives the loudspeaker in that radio may be ten volts (10V).

The ratio of output signal to input signal (measured in the same units) of a system is referred to as its **gain**.

In the example above, the gain of the radio receiver would be:

$$\frac{10V}{10\mu V} = 1{,}000{,}000$$

Because the units in the equation are those of volts, this gain would be referred to as the **voltage gain** of the receiver.

There are other types of gain that could be quoted for the above receiver. A typical input impedance (resistance) for a radio receiver is 50Ω. Using Ohm's Law, the power applied to that input when 10μV is applied can thus be calculated to be $2x10^{-12}$W and the power output to the speaker to be 6.25W if the loudspeaker's resistance is 16Ω. The **power gain** of the receiver is thus given by:

$$\frac{6.25W}{2x10^{-12}W} = 3.125x10^{12}$$

This is a huge number! More on this later.

Now to calculate the signal currents at the input and the output of the receiver. Again using Ohm's Law, the input current is 0.2μA and the output current is 0.625A. The **current gain** will thus be:

$$\frac{0.625A}{0.2\mu\text{A}} = 3{,}125{,}000$$

Attenuation

For some electronic systems the magnitude of the output signal will be smaller than that of the input signal.

As an example, consider a long telephone line. Due to the resistance of the wire from which the line is constructed there will be loss in the line and the magnitude of the signal at the telephone end of the line (the subscriber's end) will be lower than that at the originating end of the line. It is quite possible for the signal power to be halved in magnitude by the losses in a line. This gives a gain of 0.5 for the line.

Thus gains of less than unity signify **loss** in a system. The term used for this loss is **attenuation**.

The decibel

As can be seen from the two examples of systems given above, the gain of an electronic system may have a value anywhere between a small fraction (if the loss is high) to a huge number (as in the case of the radio receiver).

The use of such a large range of numbers to quantify the gain of systems is, in many ways, very inconvenient. To make things easier, the logarithm of the ratio of output to input is often calculated and quoted.

For example, the logarithm of the power gain of the radio receiver is:

$$\log_{10} 3.125\text{x}10^{12} = 12.5$$

and the logarithm of the gain of the telephone line is:

$$\log_{10} 0.5 = -0.3$$

These are much more convenient numbers to deal with!

Notice that using logs gives positive numbers for gain and negative numbers for loss (attenuation).

The idea of using the log of the ratio was developed in the 19^{th} century to describe the losses associated with long telephone lines. It was initially called the 'transmission unit' and was given the unit name 'the bel', in honour of the telephone pioneer Alexander Graham Bell. However, the bel was rather too large a unit for easy practical use, so the numbers were multiplied by ten and the unit '**decibel**' used.

Thus, this gives the power gain of the radio receiver as:

$$10\log_{10} 3.125\text{x}10^{12} = 125 \text{ decibels}$$

and the power gain of the telephone line as:

$$10\log_{10} 0.5 = -3 \text{ decibels.}$$

These would normally be written as 125dB and –3dB, respectively.

Again, note that positive dBs mean gain and negative dBs mean attenuation.

The units 'bel' or 'decibel' are defined using the ratio of the output to input **power** of a system. Now see what happens if a voltage, or a current ratio is used.

$$\text{Power Gain} = 10\log_{10} \frac{P_{out}}{P_{in}} \text{dB}$$

Now: $P = V^2/R$, therefore

$$\text{Power Gain} = 10\log_{10}\frac{\frac{V_{out}^2}{R_L}}{\frac{V_{in}^2}{R_{in}}} = 20\log_{10}\frac{V_{out}}{V_{in}}\ \text{dB (if } R_L = R_{in}\text{)}$$

So, using a voltage ratio means that you have to use 20 times, instead of 10, to get the correct answer in decibels. You will see later what happens if $R_L \neq R_{in.}$

Also, because $P = I^2R$, a similar result is achieved if a current ratio is used, giving:

$$\text{Power Gain} = 20\log_{10}\frac{I_{out}}{I_{in}}\ \text{dB}$$

Now consider a system comprising two parts, the first of which has a power gain of 3 and the second a power gain of 6. To get the total power gain of the system you need to **multiply** the two gains of the parts, giving 18.

Now, in decibels the gains are:

$$10\log_{10} 3 = 4.77\ \text{dB}$$

$$10\log_{10} 6 = 7.78\ \text{dB}$$

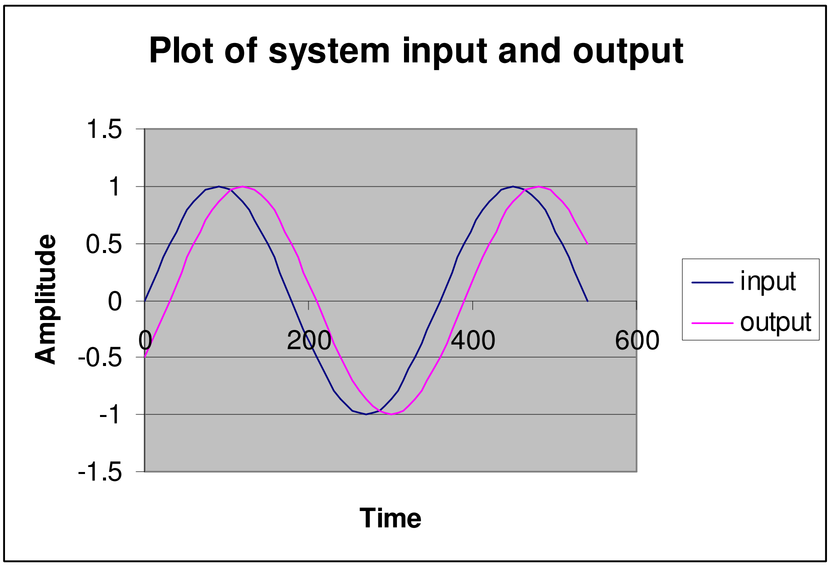
$$10\log_{10} 18 = 12.55\ \text{dB}$$

Notice that to get the total gain in dB you just **add** the individual gains in dB – much easier to do than multiplication!

Phase

A second reason for the output of an electronic system to be identical to its input is one of time delay. It takes time for the signals to pass through the system. This time delay may only be parts of a microsecond, but it can have a considerable effect on system performance.

Consider a sinusoidal input signal to a system and the corresponding delayed output (for this example, and simplicity, it is assumed that the system has unity gain). The input and output waveforms may look like the diagram below.



You will see that the output waveform is delayed by a small amount with respect to the input waveform. The delay could be measured in units of time but it is more usual to express it as an angle. This can be done because one cycle of the waveform is equivalent to 2п radians (360 degrees). In the diagram the difference between the two waveforms is 30 degrees.

The difference in degrees (or radians) is referred to as the **phase shift** (or just phase) between the two waveforms.

Frequency Effects

In an electronic system gain and phase are seldom constant with respect to the frequency of the applied signal. Because of this, the system is said to have a **frequency response**. This is just a mathematical, or pictorial (graph) description of how the gain and phase of the system change with frequency.

The frequency response of a system is an important property of that system. Some systems give an increase in gain with increasing frequency. Such a response is called a **high-pass** response.

Some systems have a frequency response with the gain dropping as the frequency increases. Such a response is called a **low-pass** response.

In some systems the gain increases with frequency up to some value and then decreases as the frequency is further increases. Such a response is called a **bandpass** response.

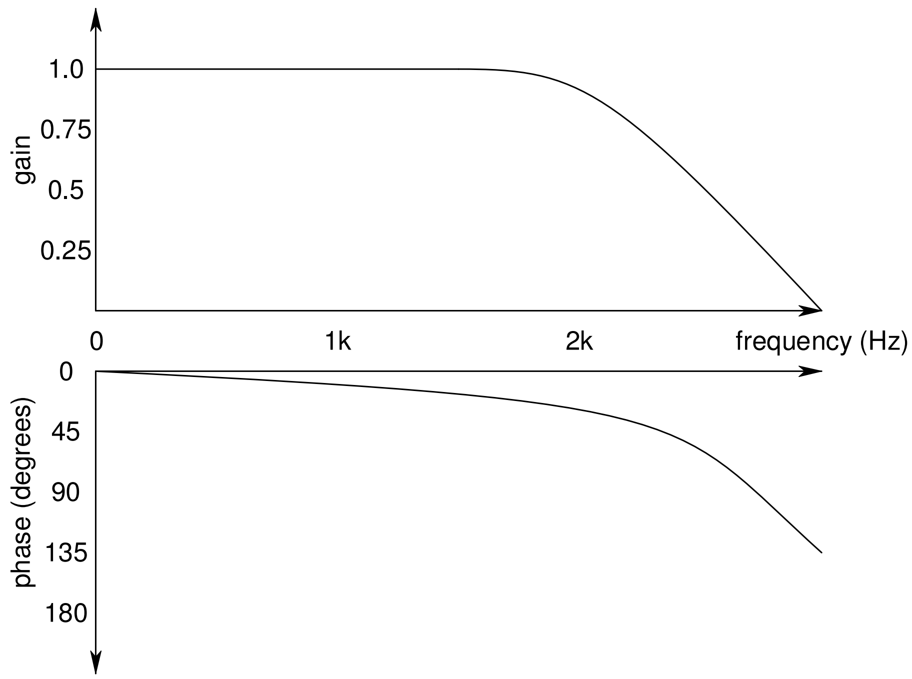
You will be meeting systems with these types of response as you progress through your course.

Plots

One of the most convenient ways of describing the frequency response of a system is in graphical form. This is usually not as accurate as describing it mathematically, but it is often adequate for practical purposes and is normally much easier to see what is happening from a graph, rather than from the mathematics.

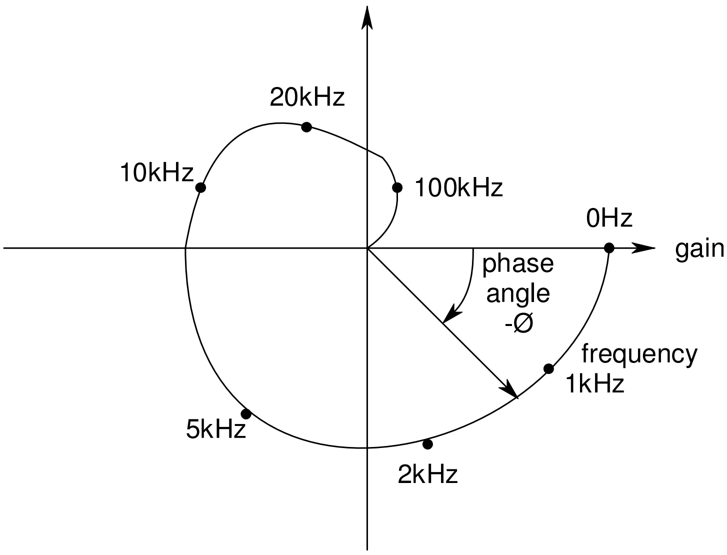
There are two main forms of graphs that are used to show the frequency response of a system. These are named after the persons that devised them and are the **Bode** plot and the **Nyquist** plot.

The **Bode** plot of a system is no more than normal graphs of gain (on the Y axis) against frequency (on the X axis) together with another graph of phase (on the X axis) against frequency. Generally, the gain and the phase curves are plotted on separate axes, one above the other, as shown in the diagram below. The frequency axes are the same for both graphs, so a direct relationship between gain and phase at any required frequency can be made easily.



The second way of displaying the frequency response is by using a vector (or phasor) type plot in which the gain of the system is given by the length of the vector and the phase by the angle of the vector. This type of plot is known as a **Nyquist** plot, after the mathematician who devised it.

An example Nyquist plot of a system is given below.



As you can see, the gain and phase at different frequencies are given by the length and angle (Φ) of the phasor. The corresponding frequencies are usually shown on the plot along the **locus** (the path) of the curve.

Other names for the plots are:

Bode plot – **rectangular** plot

Nyquist plot – **polar** plot

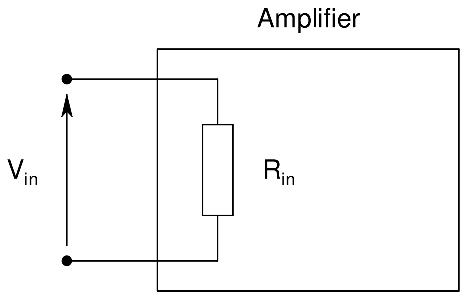
The type of plot that is used depends on the type of system that is being investigated and the properties of the system under investigation. For example, the frequency response of amplifiers and filters normally use the Bode plot form of graph, whereas investigations into control systems and stability use the Nyquist form. However, it is not incorrect to use either form.

You will use both types of plot as you perform assignments using this equipment.

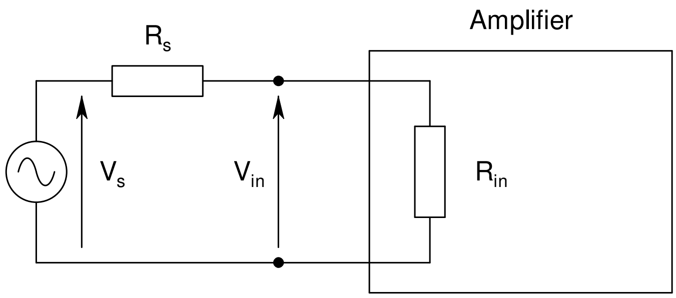
Determining Input and Output Resistances of Voltage Amplifiers

Input Resistance

The input resistance of an amplifier can be represented by a resistance R_{in} connected internally across its input terminals, as shown in the diagram below.



If a signal voltage source is connected across the input terminals the circuit becomes:

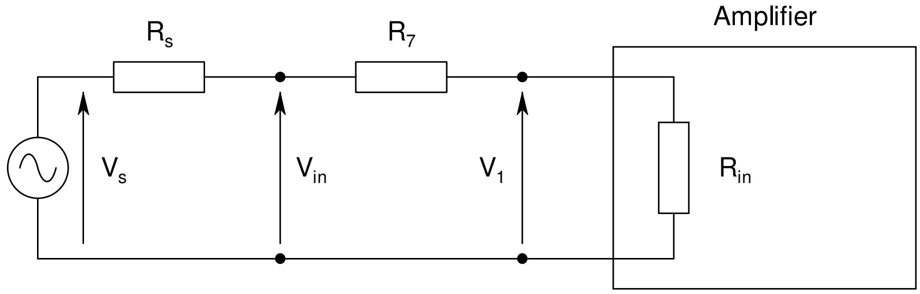


You can now see that there will be a potential divider circuit comprising the resistors R_s and R_{in} associated with the input circuit. The relationship between V_s and V_{in} will be determined by the relative values of the two resistors and can be calculate using Ohm's law.

V_s is the open circuit voltage produced by the signal source. You can determine this by just measuring the source voltage with no circuit connected to the source.

R_s is the resistance of the source. The source supplied on the workboard (the Sweep Source) has been designed to have a relatively low resistance output. However, you do not know exactly its value. So how can you determine R_{in}?

Suppose a much larger value resistor was connected in series with the source. The circuit now becomes:



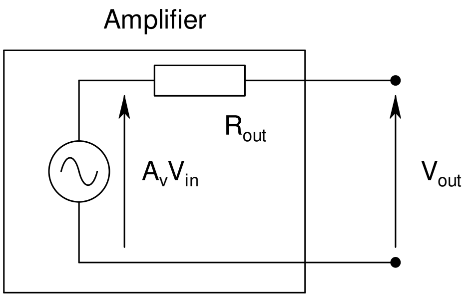
Now, both V_{in} and V_1 can be measured and, if the value of R_7 is known, the value of Rin can be determined using the normal potential divider formula:

$$\frac{V_1}{V_{in}} = \frac{R_{in}}{R_7 + R_{in}}$$

The value of the resistance R7 on the workboard is 10kΩ.

Output Resistance

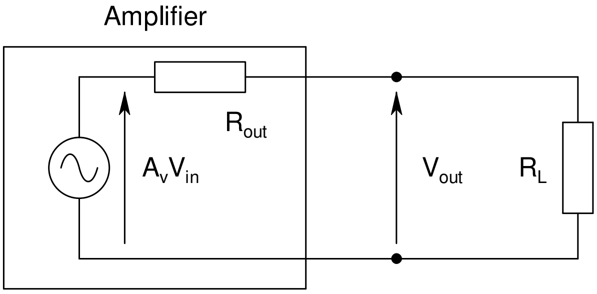
A somewhat similar technique can be used to determine the output resistance of the amplifier. The output circuit of the amplifier can be represented as an equivalent voltage source that has a value dependent on the input voltage, V_{in}, and the voltage gain of the amplifier, A_v. This voltage source is in series with the output resistance of the amplifier, R_{out}. This is shown in the following diagram.



V_{out} is the output voltage with nothing connected to the output terminals – the open circuit output voltage.

Because there is nothing connected across the output there will be no current flowing in R_{out}. So the voltage drop across R_{out} will be zero. Thus V_{out} will be the same as A_vV_{in}. You can measure this value (call it V_{out1}).

If a load resistor is now connected across the output the circuit becomes:



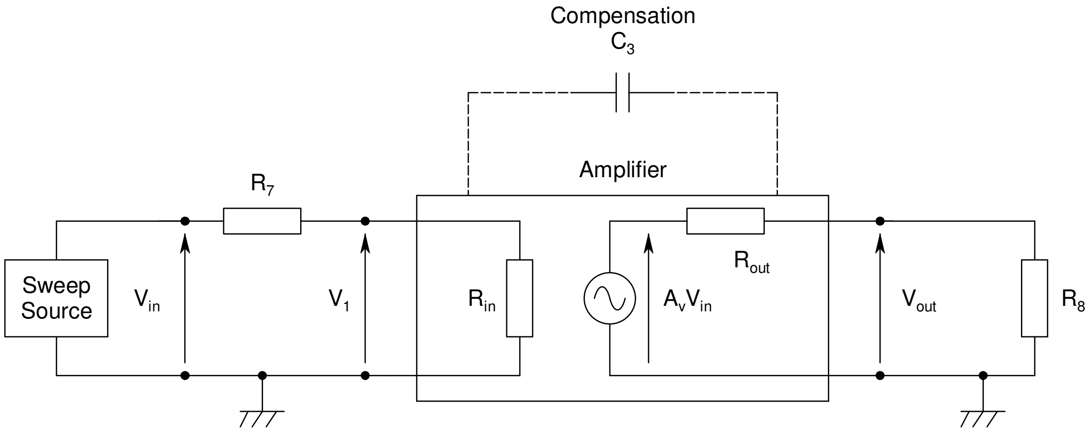
There will now be current flowing in the output circuit and thus there will be a voltage drop across R_{out}. So V_{out} will be lower than before. You can measure this new value of V_{out} (call it V_{out2}).

If R_L is known, R_{out} can be determined using a potential divider relationship again:

$$\frac{V_{out1}}{V_{out2}} = \frac{R_{out} + R_L}{R_L}$$

R_L is shown as R_8 on the workboard and the value of R_8 is 220Ω.

The full equivalent circuit representing the block on the workboard thus becomes:



The function of the compensation capacitor C_3 will become evident when you perform the Practical.

Practical 1: Voltage Amplifier Gain and Phase Characteristic

Objectives and Background

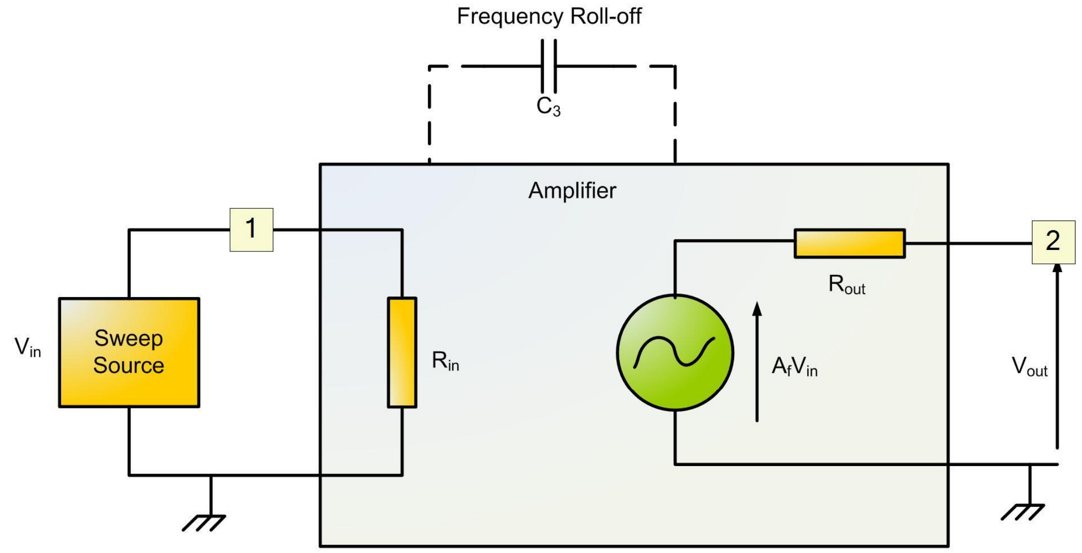
The amplifier that you will use in this Assignment is one that requires an input voltage signal and produces an output voltage signal.

In this practical you will apply a sinusoidal voltage input to the Voltage Amplifier on the workboard and, initially, you will use the oscilloscope to see how the voltage gain of the amplifier varies with frequency.

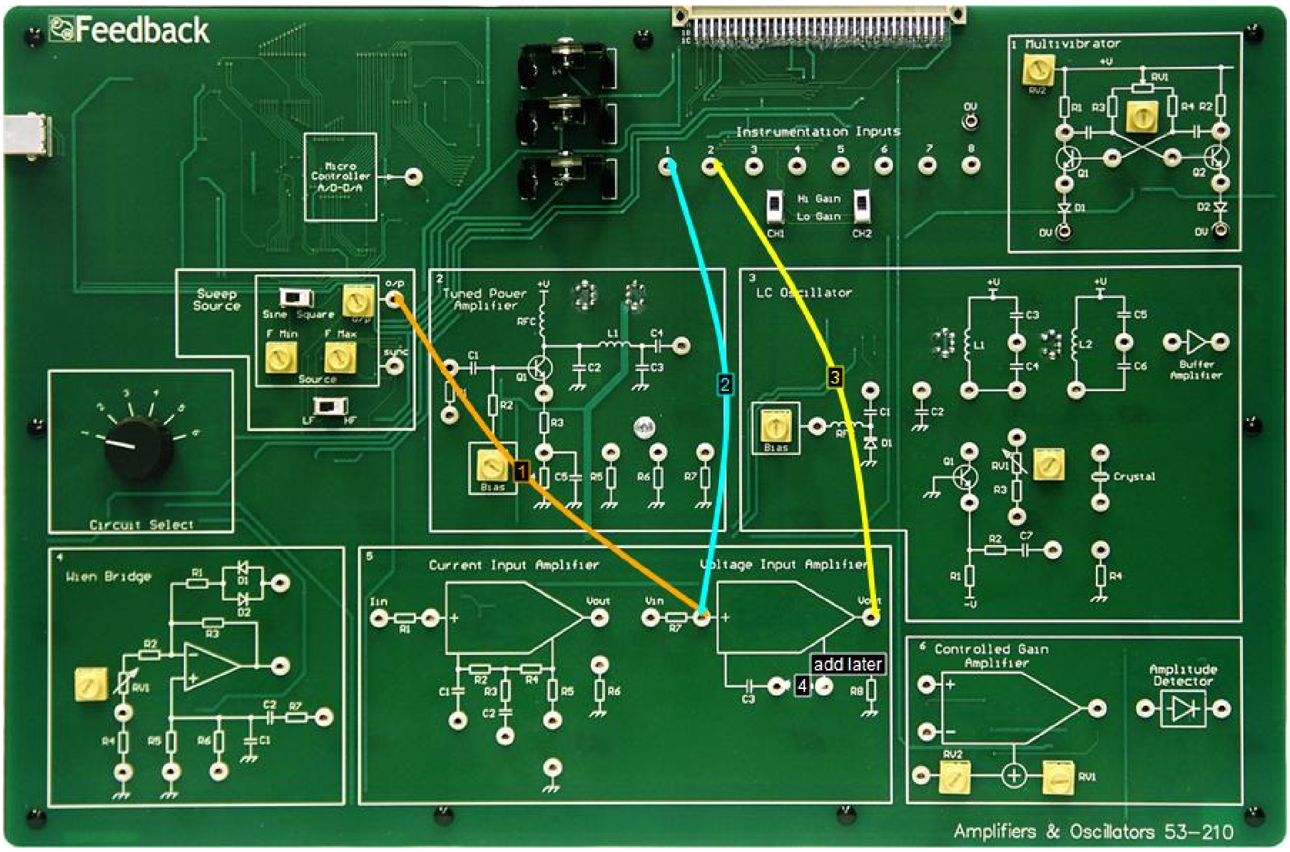
You will then use the Gain/Phase Analyser test instrument (GPA) to plot the voltage gain response automatically. You will also use the GPA to show how the phase shift through the amplifier varies with frequency.

You will see how the frequency response can be modified by the addition of a frequency roll-off capacitor.

Block Diagram



Make Connections Diagram



Practical 1: Voltage Amplifier Gain and Phase Characteristic

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that you have the **Circuit Select** switch set to **5**.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard (directly below the **Instrumentation Input** sockets) and set CH1 to **Lo Gain**, CH2 to **Hi Gain**.

Identify the **Sweep Source** circuit block at the left-hand centre of the workboard. In this block, set the **Sine/Square** switch to Sine and the **LF/HF** switch (just below the Source block) to HF. Set both the **FMin** and **FMax** source controls to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Open the oscilloscope and use the **o/p** control on the sweep source to set the its output amplitude (yellow probe on monitor point 1) to approximately 0.8V pk-pk.

Open the frequency counter and monitor the frequency of the sweep source.

Use the cursors on the input and output traces of the oscilloscope to measure their positive and negative peak voltages. Calculate the peak-to-peak input and output voltages for the amplifier and thus determine the amplifier gain (as both a ratio and in dB).

Look at the input and output waveforms and estimate the phase shift (if any) between them.

Use the FMin source control to vary frequency of the source and note how the input and output amplitudes change with frequency.

Use the FMin source control to set the frequency to 2.5MHz.

As before, use the cursors on the input and output traces of the oscilloscope to measure their positive and negative peak voltages. Calculate the peak-to-peak input and output voltages for the amplifier and thus determine the amplifier gain (as both a ratio and in dB). Also, estimate the phase shift between them.

You should have found that the gain of the amplifier is lower at the high frequency and that, at the higher frequency, there is a phase shift between the input and the output waveforms, due to the delay through the amplifier.

Close all the test equipment and open the Gain Phase Analyser (GPA). The **Set Min Freq** button on the GPA is already selected by default. Use the FMin control of the Sweep Source to set the minimum sweep frequency to approximately 100kHz. Select the **Set Max Freq** button on the GPA. Use the FMax control of the Sweep Source to set the maximum sweep frequency to approximately 8MHz.

Now, click the **Plot** button on the GPA.

The GPA automatically gives you the Bode plot of the amplifier, showing the two traces: magnitude and phase between these frequencies. Note the variations with frequency.

Use the cursor on the GPA display to measure the gain and phase at 100kHz and at 2.5MHz. Compare these measurements with those achieved using the oscilloscope method. You may now appreciate the ease of use of the GPA compared with the oscilloscope for this purpose!

Add the capacitor **C3** as instructed by the Make Connections diagram. This capacitor should alter the high frequency response of the amplifier.

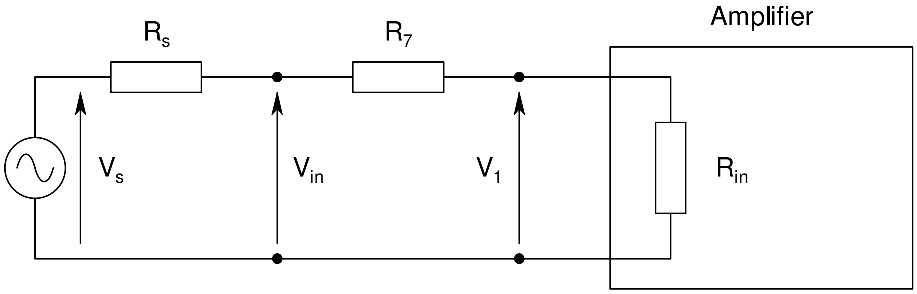
Use the GPA to give the Bode plot of the modified circuit. Note the plot and then recalibrate the GPA for a maximum frequency of approximately 2MHz.

Plot the response and compare the plots for the circuit with and without C3.

Practical 2: Measuring Input Resistance

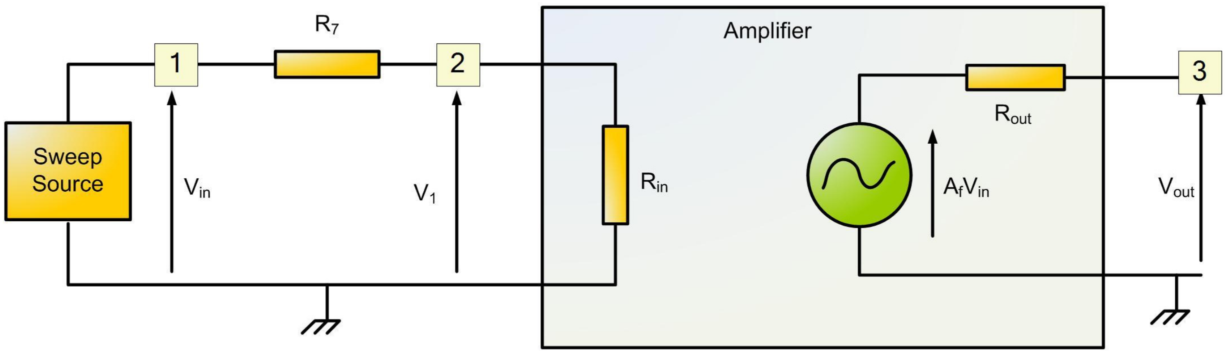
Objectives and Background

In this Practical you will use the oscilloscope and the voltmeter to measure the input signal voltage on either side of a resistor (R_7) introduced in series with the input to the amplifier.

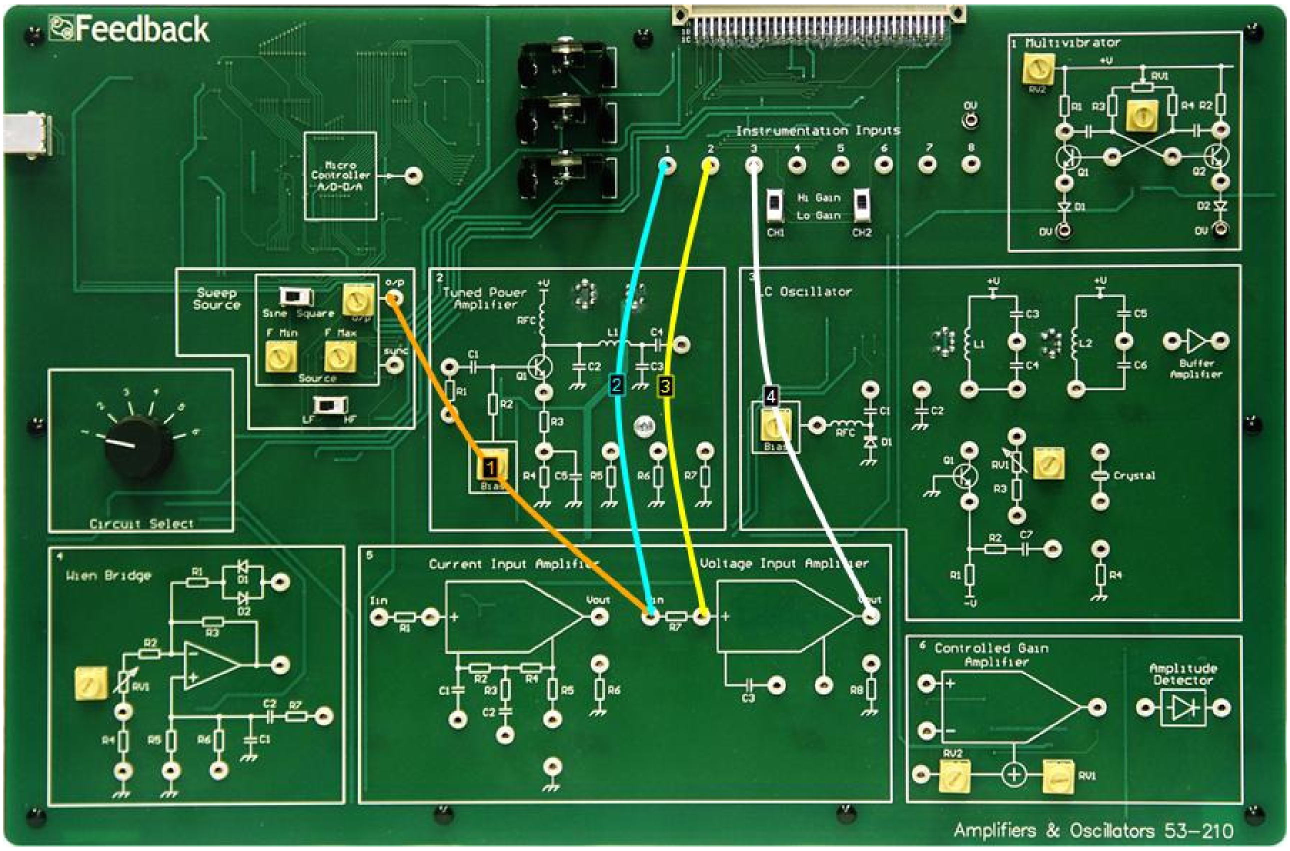


You will then use the standard potential divider formula to calculate the value of the input resistance of the amplifier.

Block Diagram



Make Connections Diagram



Practical 2: Measuring Input Resistance

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that you have the **Circuit Select** switch set to **5**.

Set **CH1** to **Lo Gain** and **CH2** to **Hi Gain**.

In the **Sweep Source** block, set the **LF**/**HF** switch to HF and the **Sine**/**Square** switch to Sine. Set both the **FMin** and **FMax** controls to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Open the oscilloscope and set the output amplitude from the Sweep Source (using the **o**/**p** control) to be approximately 0.8V pk-pk.

Open the frequency counter and use it to monitor the signal frequency.

Open the voltmeter and use it to measure the ac peak to peak amplitude of the input signal to the amplifier at both sides of $\mathbf{R_7}$ (move the yellow probe between monitor points 1 and 2).

Calculate R_{in} (knowing that $R_7 = 2.7k\Omega$).

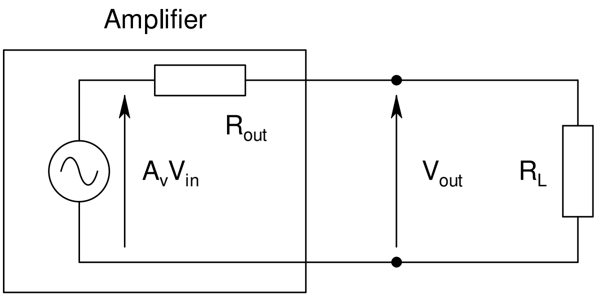
Repeat the above procedure for input frequencies of approximately 1MHz and 3MHz.

Try to explain why the result is different at each frequency.

Practical 3: Measuring Output Resistance

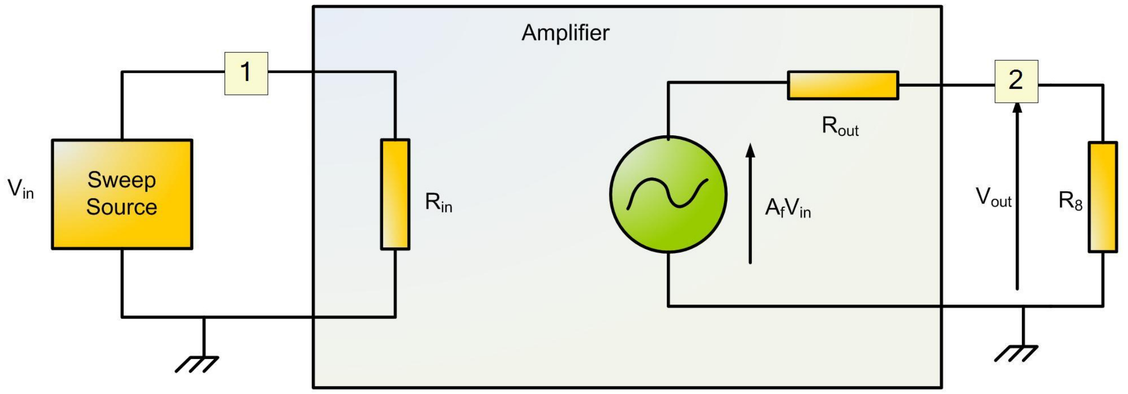
Objectives and Background

In this Practical you will use the oscilloscope and the voltmeter to measure the output signal voltage both with and without a load resistor (R_L) connected to the output of the amplifier.

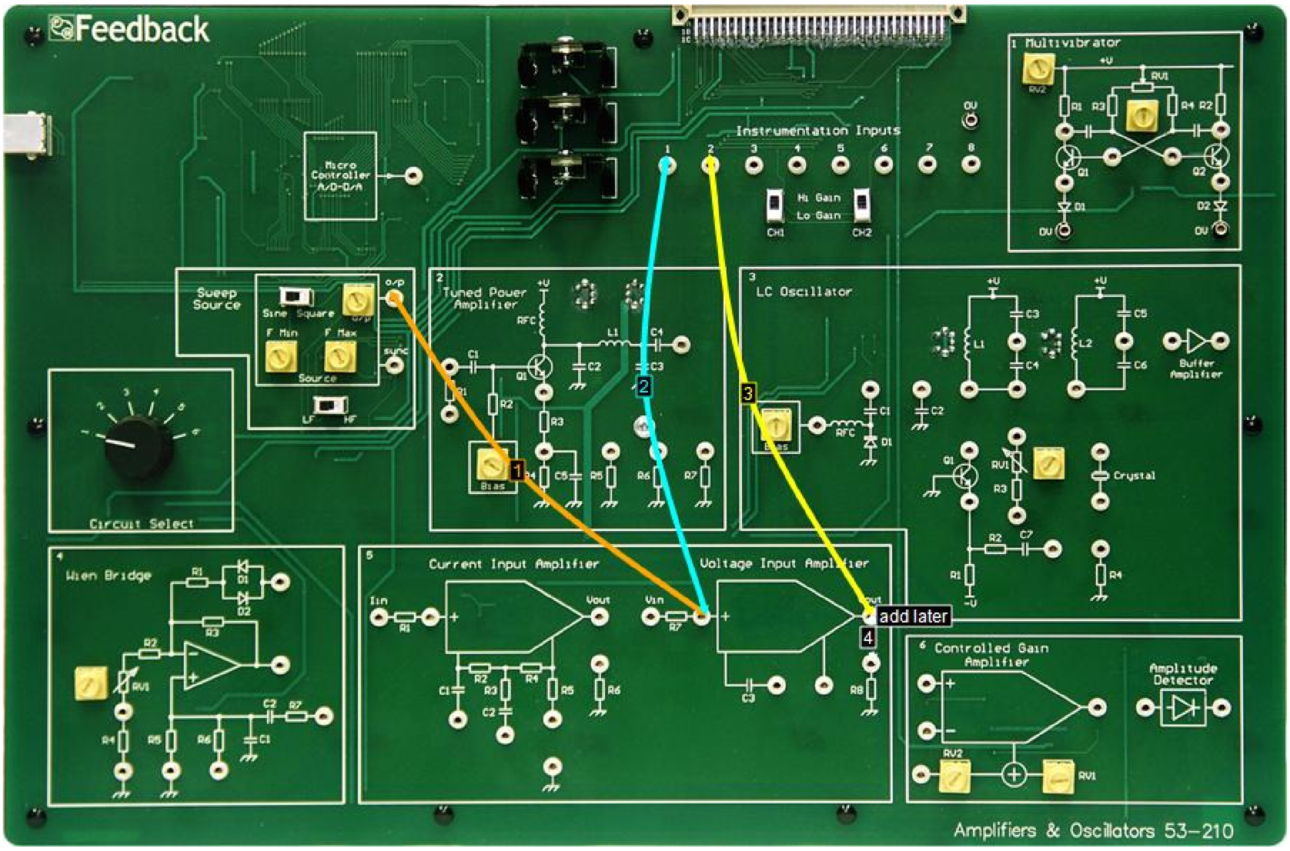


You will then use the standard potential divider formula to calculate the value of the output resistance of the amplifier.

Block Diagram



Make Connections Diagram



Practical 3: Measuring Output Resistance

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that you have the **Circuit Select** switch set to **5**.

Set **CH1** to **Lo Gain** and **CH2** to **Hi Gain**.

In the **Sweep Source** block, set the **LF**/**HF** switch to HF and the **Sine**/**Square** switch to Sine. Set both the **FMin** and **FMax** controls to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Open the oscilloscope and set the output amplitude from the Sweep Source (using the **o**/**p** control) to be approximately 0.8V pk-pk.

Open the frequency counter and use it to monitor the signal frequency.

Open the voltmeter and use it to measure the ac peak to peak amplitude of the output signal from the amplifier (the blue probe on monitor point 2).

Now connect the load resistor (R_8), as shown by connection 4 in the Make Connections diagram, and measure the new output voltage.

Calculate R_{out} (knowing that $R_8 = 220\Omega$).

Repeat the above procedure for input frequencies of approximately 1MHz and 3MHz.

Try to explain why the results are approximately the same at each frequency.

Current Input Amplifier

Objectives

To appreciate the concept of a current input, voltage output amplifier (transresistance amplifier)

To investigate the gain and the phase shift associated with such an amplifier

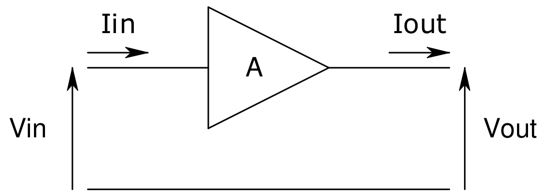
To determine the typical input resistance of such an amplifier

To determine the typical output resistance of such an amplifier

Amplifiers

Gain

The block diagram of a general amplifier is given below.



The gain of the amplifier is denoted by the symbol A. The definition of A is given by:

$$\text{Gain, A} = \frac{(output\ power)}{(input\ power)}$$

This is called the **power gain** of the amplifier.

If a circuit does not have power gain, then it is not an amplifier!

As you can see, there is present input current and input voltage and, at the output, the corresponding output current and voltage. As power is the product of current and voltage, this gives the expression for the gain:

$$A = \frac{Vout.Iout}{Vin.Iin}$$

Now:

$$\frac{Vout}{Vin} = A_V$$, the **voltage gain** of the amplifier

and:

$$\frac{Iout}{Iin} = A_I$$, the **current gain** of the amplifier

Therefore:

$$A = A_V \,.\, A_I$$

Note: that it is quite possible for an amplifier to have a voltage gain of less than one; however, its corresponding current gain must be high enough to give a power gain greater

than unity for it to be classed as an amplifier. This also works the other way: if the current gain is less than one, the voltage gain must be high enough.

Going back to the expression for gain of:

$$A = \frac{Vout.Iout}{Vin.Iin}$$

This can be further re-arranged by defining two more relationships:

$\frac{Vout}{Iin} = G_m$, the **transresistance gain** of the amplifier

and:

$\frac{Iout}{Vin} = R_m$, the **transconductance gain** of the amplifier

Giving, therefore:

$$A = G_m \,.\, R_m$$

Using a similar argument to before, to qualify as an amplifier, it is quite possible for either one or the other of these terms to be less than one providing the product of the two is greater than unity.

Types of Amplifier

The input signal to an amplifier may be a current or it may be a voltage. Therefore, this gives rise to two types of amplifier: the current input amplifier and the voltage input amplifier.

Each of these types of amplifier may be further sub-divided, as each can give a current output or a voltage output.

There are, therefore, four general forms of amplifier.

Different applications require amplifiers to have different properties of amplification.

An example

Consider a fibre optic communications system with a great distance between transmitter and receiver. Because of the attenuation of light along the length of fibre it is often necessary to compensate for this loss by having 'repeater' circuits at intervals along the fibre. The purpose of these circuits is to detect the incoming light signal, convert it into an electrical signal that can be amplified and then used to drive a secondary light source that

provides a regenerated signal for transmission further down the cable.

A typical sensor for the input of such a repeater is a photodiode. This will produce a current that is proportional to the light intensity. A typical output device might be a light-emitting diode, or perhaps a laser diode, which also needs a current signal to drive it. The amplifier within the repeater must thus take the small current output from the photodiode and amplify it to drive the LED or laser.

Therefore, the amplifier required has to be a **current amplifier**. The requirements for such a circuit are:

Its input should affect the signal current from the photodiode as little as possible,

It should have output circuitry that maximises the current transfer out of the amplifier,

It should have current gain.

To achieve this, it should have **as low an input resistance as possible** and its output should look like an ideal current source (i.e. it should have **as high an output resistance as possible**).

Another example

Consider a communications receiver system. Within the receiver there are circuits that perform functions such as high frequency amplification of the signals, the selection of the required signal (filters) and frequency translation of the signal to a lower frequency (mixers). Because of current limitations on the operating speed of analogue-to-digital converters and other digital circuitry, these functions are normally performed by analogue circuits. However, it is common practice to convert high frequency signals to a much lower frequency (often a few tens of kHz) so that such functions as demodulation or decoding and the final signal processing can be done using digital techniques. DSP (digital signal processing) circuits are widely used to do this.

Typically, DSP chips require input voltage signals of a few volts amplitude. The output signal from the analogue part of the receiver system may be only tens of millivolts in amplitude. An amplifier is therefore required to ‘bridge this gap’.

Therefore, this amplifier has to be a **voltage amplifier**. The requirements for such a circuit are:

Its input should affect the signal voltage from the analogue circuitry as little as possible,

It should have output circuitry that maximises the voltage transfer out of the amplifier,

It should have voltage gain.

To achieve this, it should have **as high an input resistance as possible** and its output should look like an ideal voltage source (i.e. it should have **as low an output resistance as possible**).

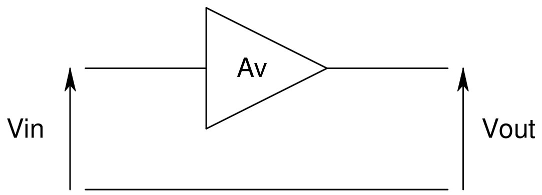
You can imagine that the requirements for the two example amplifiers above will result in completely different circuitry to satisfy them.

The two other types of amplifier also find uses in electronics and communications systems. For example, a field effect transistor amplifier is an example of a **transconductance amplifier** and **transresistance amplifiers** are widely used in audio

mixers and digital-to-analogue converters.

The Voltage Amplifier

A **voltage amplifier** is one to which an input voltage is applied and an output voltage results. The block diagram for such a system is given below.



The **voltage gain** of such an amplifier is given by

$$\text{Voltage gain,} \quad A_v = \frac{V_{out}}{V_{in}}$$

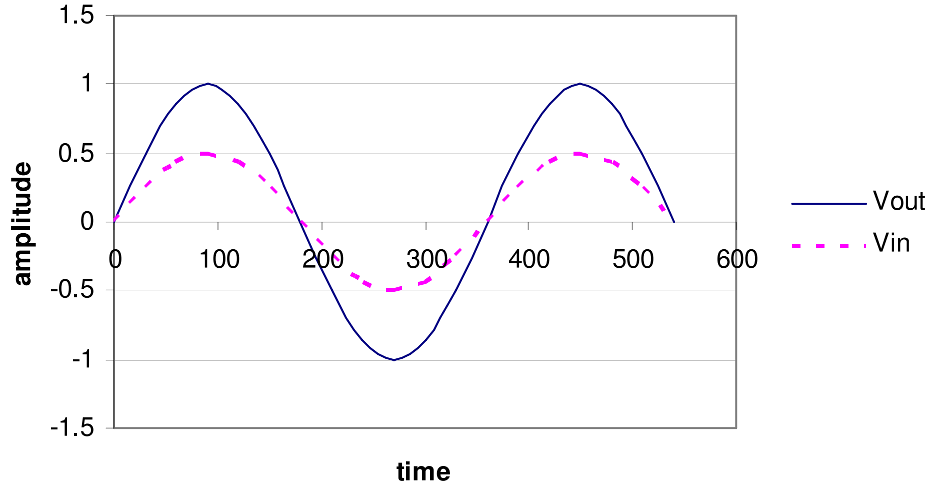
Ideally, connecting a voltage amplifier into a circuit should not affect the input voltage in any way. To achieve this, the input impedance of the amplifier should be as high as possible.

Also, ideally, connecting a load onto the output of the amplifier should not change the output voltage in any way. For this to happen, the output impedance of the amplifier should be as low as possible.

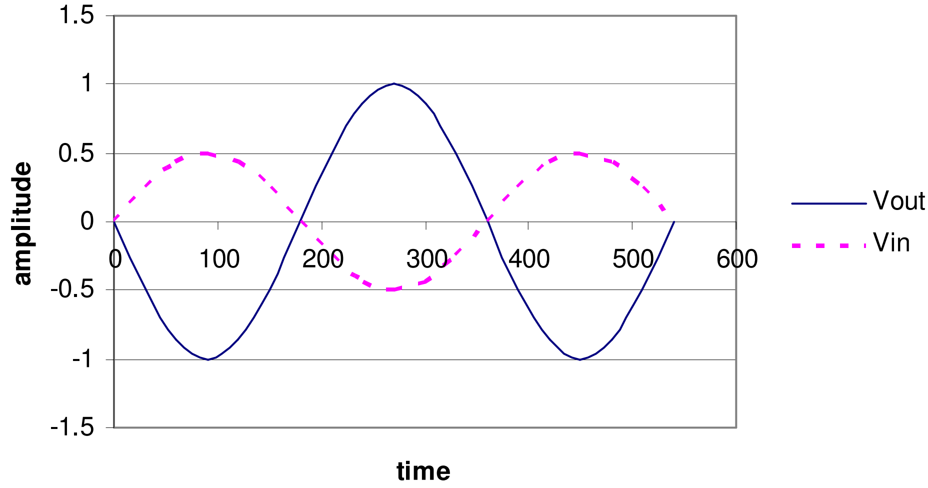
The ideal properties for a voltage amplifier are thus:

Voltage gain	Yes
Input impedance	Very high
Output impedance	Very low

The output voltage may follow the input voltage directly, or it may be inverted in polarity. This is illustrated below:



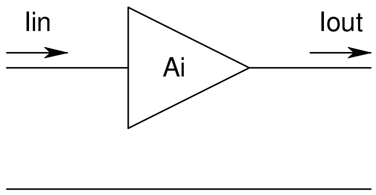
This shows the output following the input directly. The output is said to be **in phase** with the input. Another name for such an amplifier is a **non-inverting amplifier**. Note, also, that the output is amplified with respect to the input.



This shows the output inverted with respect to the input. The output is said to be in **anti-phase** with the input. Another name for such an amplifier is an **inverting amplifier**. Note, also, that the output is amplified with respect to the input.

The Current Amplifier

A **current amplifier** is one to which an input current is applied and an output current results. The block diagram for such a system is given below.



The **current gain** of such an amplifier is given by

$$\text{Current gain,}\quad A_i = \frac{I_{out}}{I_{in}}$$

Ideally, connecting a current amplifier into a circuit should not affect the input current in any way. To achieve this, the input impedance of the amplifier should be as low as possible.

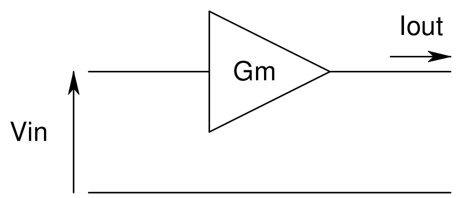
Also, ideally, connecting a load onto the output of the amplifier should not change the output current in any way. For this to happen, the output impedance of the amplifier should be as high as possible.

The ideal properties for a voltage amplifier are thus:

Current gain	Yes
Input impedance	Very low
Output impedance	Very high

The Transconductance Amplifier

A **transconductance amplifier** is one to which an input voltage is applied and an output current results. The block diagram for such a system is given below.



The **transconductance gain** of such an amplifier is given by

$$\text{Transconductance gain,} \quad G_m = \frac{I_{out}}{V_{in}}$$

Ideally, connecting a voltage amplifier into a circuit should not affect the input voltage in any way. To achieve this, the input impedance of the amplifier should be as high as possible.

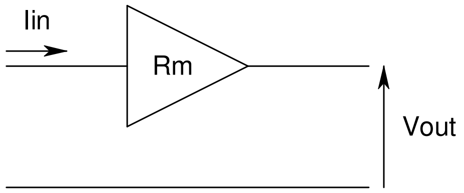
Also, ideally, connecting a load onto the output of the amplifier should not change the output current in any way. For this to happen, the output impedance of the amplifier should be as high as possible.

The ideal properties for a transconductance amplifier are thus:

Transconductance gain	Yes
Input impedance	Very high
Output impedance	Very high

The Transresistance Amplifier

A **transresistance amplifier** is one to which an input current is applied and an output voltage results. The block diagram for such a system is given below.



The **transresistance gain** of such an amplifier is given by

$$\text{Transresistance gain,} \quad R_m = \frac{V_{out}}{I_{in}}$$

Ideally, connecting a transresistance amplifier into a circuit should not affect the input current in any way. To achieve this, the input impedance of the amplifier should be as low as possible.

Also, ideally, connecting a load onto the output of the amplifier should not change the output voltage in any way. For this to happen, the output impedance of the amplifier should be as low as possible.

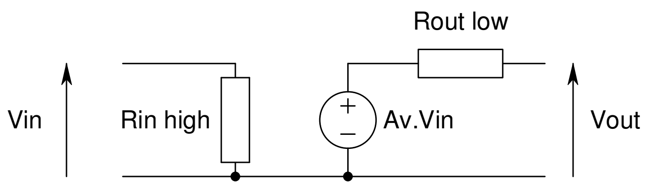
The ideal properties for a transresistance amplifier are thus:

Transresistance gain	Yes
Input impedance	Very low
Output impedance	Very low

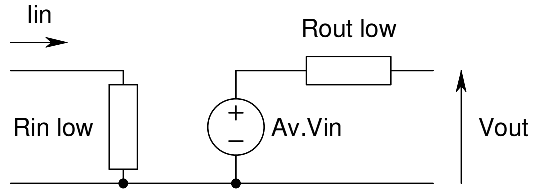
Summary of Amplifier Properties

A summary of the properties of the four types of amplifier, together with an equivalent circuit for each, is given below.

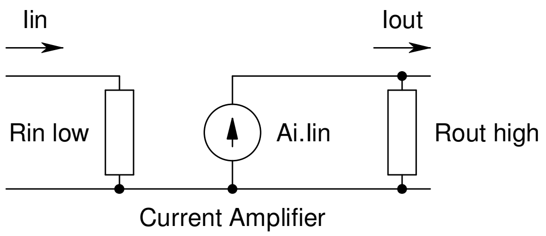
Amplifier Type	**Gain Required**	**Input Resistance**	**Output Resistance**
Voltage Amplifier	Voltage A_V	Very high	Very low
Current Amplifier	Current A_I	Very low	Very high
Transconductance Amplifier	Transconductance G_m	Very high	Very high
Transresistance Amplifier	Transresistance R_m	Very low	Very low

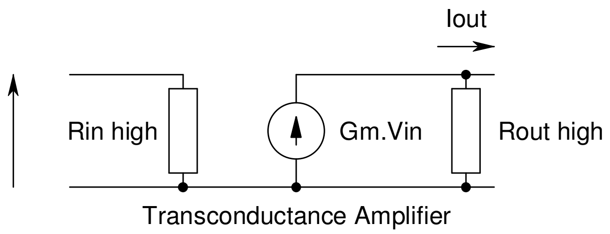


Voltage Amplifier



Transresistance Amplifier


Iin
Iout
Rin low
Ai.Iin
Rout high
Current Amplifier


Iout
Rin high
Gm.Vin
Rout high
Transconductance Amplifier

Gain and Phase

Gain

If a signal is applied to an electronic system, the output of that system is unlikely to be exactly the same as the input. For instance, the output is likely to be different in magnitude.

For example, the signal input voltage at the antenna of a radio receiver may be ten microvolts (10µV), whereas the output voltage that drives the loudspeaker in that radio may be ten volts (10V).

The ratio of output signal to input signal (measured in the same units) of a system is referred to as its **gain**.

In the example above, the gain of the radio receiver would be:

$$\frac{10V}{10\mu V} = 1{,}000{,}000$$

Because the units in the equation are those of volts, this gain would be referred to as the **voltage gain** of the receiver.

There are other types of gain that could be quoted for the above receiver. A typical input impedance (resistance) for a radio receiver is 50Ω. Using Ohm's Law, the power applied to that input when 10µV is applied can thus be calculated to be $2x10^{-12}$W and the power output to the speaker to be 6.25W if the loudspeaker's resistance is 16Ω. The **power gain** of the receiver is thus given by:

$$\frac{6.25W}{2x10^{-12}W} = 3.125x10^{12}$$

This is a huge number! More on this later.

Now to calculate the signal currents at the input and the output of the receiver. Again using Ohm's Law, the input current is 0.2µA and the output current is 0.625A. The **current gain** will thus be:

$$\frac{0.625A}{0.2\mu\text{A}} = 3{,}125{,}000$$

Attenuation

For some electronic systems the magnitude of the output signal will be smaller than that of the input signal.

As an example, consider a long telephone line. Due to the resistance of the wire from which the line is constructed there will be loss in the line and the magnitude of the signal at the telephone end of the line (the subscriber's end) will be lower than that at the originating end of the line. It is quite possible for the signal power to be halved in magnitude by the losses in a line. This gives a gain of 0.5 for the line.

Thus gains of less than unity signify **loss** in a system. The term used for this loss is **attenuation**.

The decibel

As can be seen from the two examples of systems given above, the gain of an electronic system may have a value anywhere between a small fraction (if the loss is high) to a huge number (as in the case of the radio receiver).

The use of such a large range of numbers to quantify the gain of systems is, in many ways, very inconvenient. To make things easier, the logarithm of the ratio of output to input is often calculated and quoted.

For example, the logarithm of the power gain of the radio receiver is:

$$\log_{10} 3.125 \times 10^{12} = 12.5$$

and the logarithm of the gain of the telephone line is:

$$\log_{10} 0.5 = -0.3$$

These are much more convenient numbers to deal with!

Notice that using logs gives positive numbers for gain and negative numbers for loss (attenuation).

The idea of using the log of the ratio was developed in the 19th century to describe the losses associated with long telephone lines. It was initially called the ‘transmission unit’ and was given the unit name ‘the bel’, in honour of the telephone pioneer Alexander Graham Bell. However, the bel was rather too large a unit for easy practical use, so the numbers were multiplied by ten and the unit ‘**decibel**’ used.

Thus, this gives the power gain of the radio receiver as:

$$10\log_{10} 3.125 \times 10^{12} = 125 \text{ decibels}$$

and the power gain of the telephone line as:

$$10\log_{10} 0.5 = -3 \text{ decibels.}$$

These would normally be written as 125dB and –3dB, respectively.

Again, note that positive dBs mean gain and negative dBs mean attenuation.

The units ‘bel’ or ‘decibel’ are defined using the ratio of the output to input **power** of a system. Now see what happens if a voltage, or a current ratio is used.

$$\text{Power Gain} = 10\log_{10} \frac{P_{out}}{P_{in}} \text{dB}$$

Now: $P = V^2/R$, therefore

$$\text{Power Gain} = 10\log_{10}\frac{\frac{V_{out}^2}{R_L}}{\frac{V_{in}^2}{R_{in}}} = 20\log_{10}\frac{V_{out}}{V_{in}}\,\text{dB (if } R_L = R_{in}\text{)}$$

So, using a voltage ratio means that you have to use 20 times, instead of 10, to get the correct answer in decibels. You will see later what happens if $R_L \neq R_{in}$.

Also, because $P = I^2R$, a similar result is achieved if a current ratio is used, giving:

$$\text{Power Gain} = 20\log_{10}\frac{I_{out}}{I_{in}}\,\text{dB}$$

Now consider a system comprising two parts, the first of which has a power gain of 3 and the second a power gain of 6. To get the total power gain of the system you need to **multiply** the two gains of the parts, giving 18.

Now, in decibels the gains are:

$$10\log_{10} 3 = 4.77\text{ dB}$$

$$10\log_{10} 6 = 7.78\text{ dB}$$

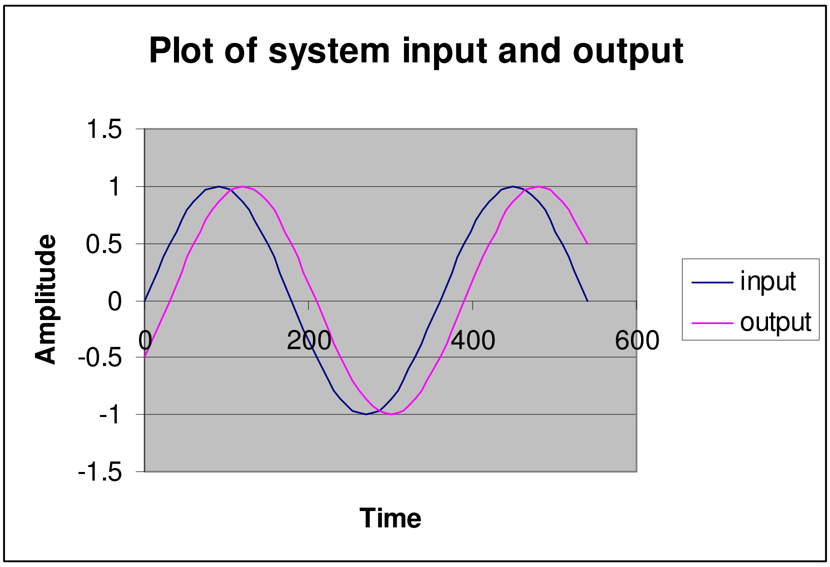
$$10\log_{10} 18 = 12.55\text{ dB}$$

Notice that to get the total gain in dB you just **add** the individual gains in dB – much easier to do than multiplication!

Phase

A second reason for the output of an electronic system to be identical to its input is one of time delay. It takes time for the signals to pass through the system. This time delay may only be parts of a microsecond, but it can have a considerable effect on system performance.

Consider a sinusoidal input signal to a system and the corresponding delayed output (for this example, and simplicity, it is assumed that the system has unity gain). The input and output waveforms may look like the diagram below.



You will see that the output waveform is delayed by a small amount with respect to the input waveform. The delay could be measured in units of time but it is more usual to express it as an angle. This can be done because one cycle of the waveform is equivalent to 2п radians (360 degrees). In the diagram the difference between the two waveforms is 30 degrees.

The difference in degrees (or radians) is referred to as the **phase shift** (or just phase) between the two waveforms.

Frequency Effects

In an electronic system gain and phase are seldom constant with respect to the frequency of the applied signal. Because of this, the system is said to have a **frequency response**. This is just a mathematical, or pictorial (graph) description of how the gain and phase of the system change with frequency.

The frequency response of a system is an important property of that system. Some systems give an increase in gain with increasing frequency. Such a response is called a **high-pass** response.

Some systems have a frequency response with the gain dropping as the frequency increases. Such a response is called a **low-pass** response.

In some systems the gain increases with frequency up to some value and then decreases as the frequency is further increases. Such a response is called a **bandpass** response.

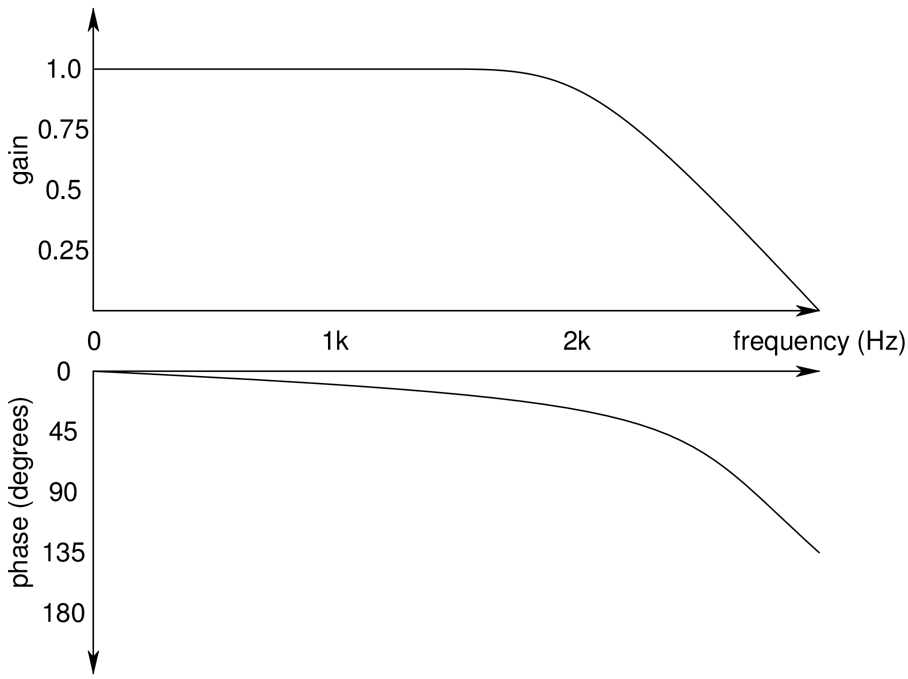
You will be meeting systems with these types of response as you progress through your course.

Plots

One of the most convenient ways of describing the frequency response of a system is in graphical form. This is usually not as accurate as describing it mathematically, but it is often adequate for practical purposes and is normally much easier to see what is happening from a graph, rather than from the mathematics.

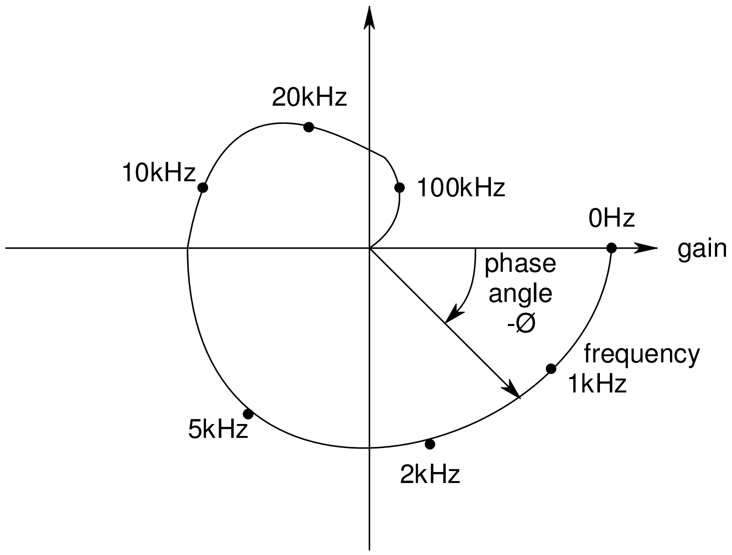
There are two main forms of graphs that are used to show the frequency response of a system. These are named after the persons that devised them and are the **Bode** plot and the **Nyquist** plot.

The **Bode** plot of a system is no more than normal graphs of gain (on the Y axis) against frequency (on the X axis) together with another graph of phase (on the X axis) against frequency. Generally, the gain and the phase curves are plotted on separate axes, one above the other, as shown in the diagram below. The frequency axes are the same for both graphs, so a direct relationship between gain and phase at any required frequency can be made easily.



The second way of displaying the frequency response is by using a vector (or phasor) type plot in which the gain of the system is given by the length of the vector and the phase by the angle of the vector. This type of plot is known as a **Nyquist** plot, after the mathematician who devised it.

An example Nyquist plot of a system is given below.



As you can see, the gain and phase at different frequencies are given by the length and angle (Φ) of the phasor. The corresponding frequencies are usually shown on the plot along the **locus** (the path) of the curve.

Other names for the plots are:

Bode plot – **rectangular** plot

Nyquist plot – **polar** plot

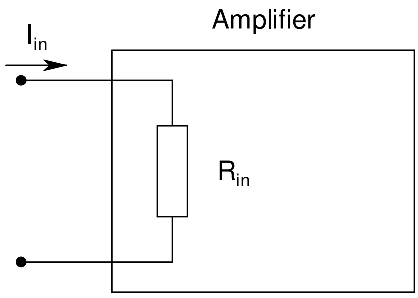
The type of plot that is used depends on the type of system that is being investigated and the properties of the system under investigation. For example, the frequency response of amplifiers and filters normally use the Bode plot form of graph, whereas investigations into control systems and stability use the Nyquist form. However, it is not incorrect to use either form.

You will use both types of plot as you perform assignments using this equipment.

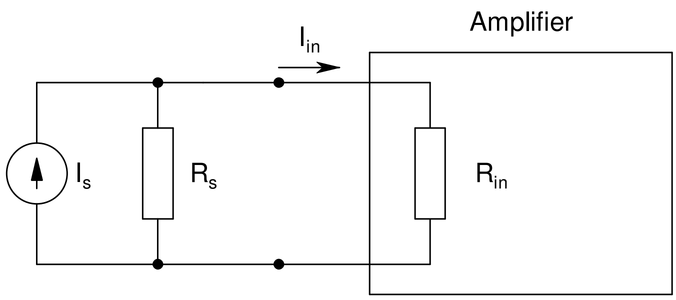
Determining Input and Output Resistances of Transresistance Amplifiers

Input Resistance

The input resistance of an amplifier can be represented by a resistance R_{in} connected internally across its input terminals, as shown in the diagram below.



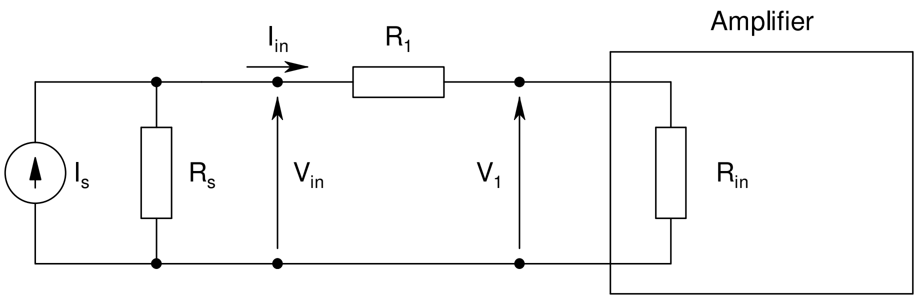
If a signal current source is connected to the input terminals the circuit becomes:



You can now see that there will be a current divider circuit comprising the resistors R_s and R_{in} associated with the input circuit. The relationship between I_s and I_{in} will be determined by the relative values of the two resistors and can be calculate using Ohm's law.

R_s is the resistance of the source. The source supplied on the workboard (the Sweep Source) has been designed to have a relatively high resistance output. However, you do not know exactly its value. So how can you determine R_{in}?

Suppose a resistor was connected in series with the source. The circuit now becomes:



The current I_{in} flows through both resistor R_1 and the input resistance R_{in}. There will be a voltage drop across R_1 due to this current.

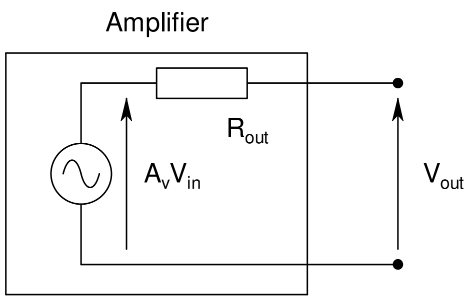
Now, both V_{in} and V_1 can be measured and, if the value of R_1 is known, the value of Rin can be determined using the normal potential divider formula:

$$\frac{V_1}{V_{in}} = \frac{R_{in}}{R_1 + R_{in}}$$

The value of the resistance R_1 on the workboard is 10kΩ.

Output Resistance

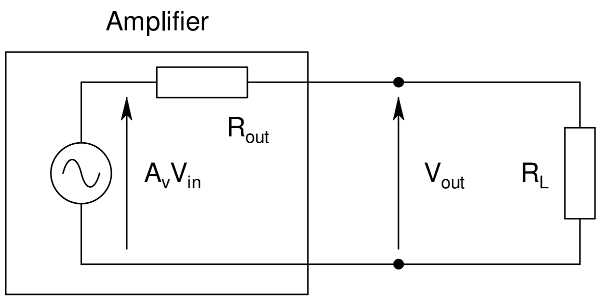
A somewhat similar technique can be used to determine the output resistance of the amplifier. The output circuit of the amplifier can be represented as an equivalent voltage source that has a value dependent on the input voltage, V_{in}, and the voltage gain of the amplifier, A_v. This voltage source is in series with the output resistance of the amplifier, R_{out}. This is shown in the following diagram.



V_{out} is the output voltage with nothing connected to the output terminals – the open circuit output voltage.

Because there is nothing connected across the output there will be no current flowing in R_{out}. So the voltage drop across R_{out} will be zero. Thus V_{out} will be the same as A_vV_{in}. You can measure this value (call it V_{out1}).

If a load resistor is now connected across the output the circuit becomes:



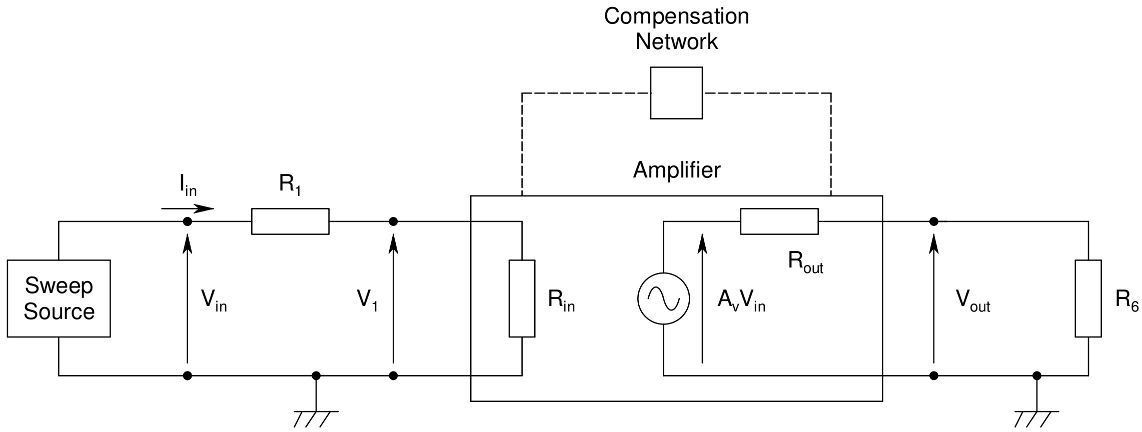
There will now be current flowing in the output circuit and thus there will be a voltage drop across R_{out}. So V_{out} will be lower than before. You can measure this new value of V_{out} (call it V_{out2}).

If R_L is known, R_{out} can be determined using a potential divider relationship again:

$$\frac{V_{out1}}{V_{out2}} = \frac{R_{out} + R_L}{R_L}$$

R_L is shown as R_6 on the workboard and the value of R_6 is 220Ω.

The full equivalent circuit representing the block on the workboard thus becomes:



The function of the compensation network will become evident when you perform the Practical.

Gain, decibels and Input and Output Resistances

The following section uses some mathematics to explain how the decibel is used to describe the gain of an amplifier. Although it may look complex, working through it should make things much clearer. There is absolutely no need to commit these relationships to memory or be able to reproduce them. The important point is that you appreciate that this apparently complex concept is based on nothing more than Ohm's law and logarithms.

In the Introduction on Gain and phase, the power gain of a system in decibels was defined as:

$$10\log_{10}\frac{P_{out}}{P_{in}}\ \text{dB}$$

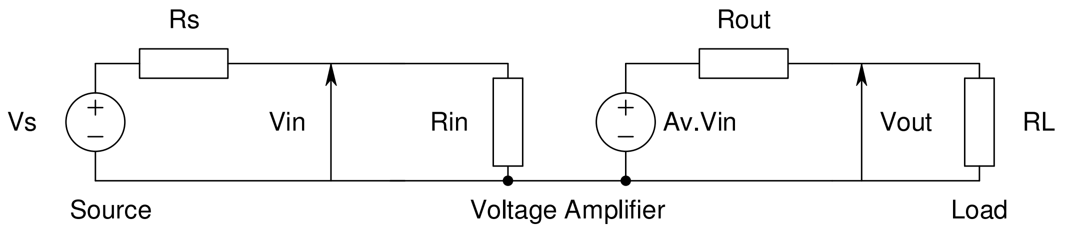
Expressing gain in dB should essentially be confined to the power gain, as the definition of the dB relates to power only. However, it was shown that the above expression became:

$$20\log_{10}\frac{V_{out}}{V_{in}}\ \text{dB} \qquad \textbf{if } \mathbf{R_L = R_{in}}.$$

So, in practice, the voltage gain of an amplifier is often expressed in dB, using this expression. Unfortunately, users often do not appreciate that normally R_L and R_{in} are normally **not** equal and that this inequality affects the gain.

By investigating the equivalent circuit of the amplifier, including its source and load, you can now see what happens if R_L and R_{in} are not equal.

The full equivalent circuit of a voltage amplifier, together with source and load, is given below.



From this:

$$P_{in} = \frac{V_{in}^2}{R_{in}} \text{ and } P_{out} = \frac{V_{out}^2}{R_L}$$

Therefore:

$$\text{Power gain} = \frac{P_{out}}{P_{in}} = \frac{V_{out}^2}{V_{in}^2}\cdot\frac{R_{in}}{R_L}$$

Expressing this is dB gives:

$$\text{Power gain} = 10\log_{10}\frac{P_{out}}{P_{in}} = 10\log_{10}\frac{V_{out}^2}{V_{in}^2}\cdot\frac{R_{in}}{R_L} \quad \text{dB}$$

$$= 20\log_{10}\frac{V_{out}}{V_{in}}\cdot + 10\log_{10}\cdot\frac{R_{in}}{R_L} \quad \text{dB}$$

From this expression you can see that, for a voltage amplifier, if the input and load resistances are **not** equal, the power gain in dB is **no longer** just 20 times the log of the voltage gain. There is the second term that is dependent on the ratio of those resistances.

If $\cdot\frac{R_{in}}{R_L} = 1$, then $10\log_{10}\cdot\frac{R_{in}}{R_L}$ will be zero and the power gain in dB will be the same as the voltage gain in dB.

If $\cdot\frac{R_{in}}{R_L} < 1$, then $10\log_{10}\cdot\frac{R_{in}}{R_L}$ will be negative and the power gain in dB will be less than the voltage gain in dB.

If $\cdot\frac{R_{in}}{R_L} > 1$, then $10\log_{10}\cdot\frac{R_{in}}{R_L}$ will be positive and the power gain in dB will be higher than the voltage gain in dB.

This shows that the gain of an amplifier depends on its input and load resistances. This is an important point to understand.

A similar analysis can be done for the other types of amplifier. The equations that result are given below.

Current amplifier:

$$\text{Power gain} = 20\log_{10}\frac{I_{out}}{I_{in}}\cdot + 10\log_{10}\cdot\frac{R_L}{R_{in}} \quad \text{dB}$$

$$\text{i.e. Power gain} = 20\log_{10} \text{Ai} + 10\log_{10}\cdot\frac{R_L}{R_{in}} \quad \text{dB}$$

Transconductance amplifier:

$$\text{Power gain} = 20\log_{10}\frac{I_{out}}{V_{in}}\cdot + 10\log_{10} R_L.R_{in} \quad \text{dB}$$

$$\text{i.e. Power gain} = 20\log_{10} G_m + 10\log_{10} R_L.R_{in} \quad \text{dB}$$

Transresistance amplifier:

$$\text{Power gain} = 20\log_{10}\frac{V_{out}}{I_{in}}\cdot + 10\log_{10}\frac{1}{R_L.R_{in}} \quad \text{dB}$$

i.e. Power gain = $20\log_{10} R_m + 10\log_{10} \frac{1}{R_L.R_{in}}$ dB

These also show that the gain of an amplifier depends on its input and load resistances.

Thus the dB value of the power gain and that of the voltage gain (or current, transconductance or transresistance gain, depending on the type of amplifier) will only be equal when the input and load resistances are equal.

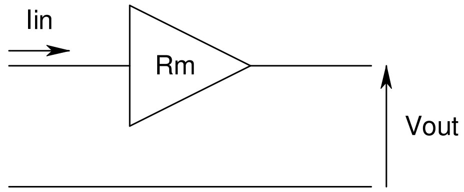
As you can now see, stating the voltage or current gain of an amplifier in decibels without some knowledge of the input resistance and the output load can well be misleading.

However, the use of $20\log_{10} \frac{V_{out}}{V_{in}}$ has been adopted in practice as a convenient definition of the voltage gain of an amplifier (in dB), regardless of the magnitudes of the input and output resistances. This is a case of engineering "slang", and worse, is often done by those who do not appreciate how misleading it can be.

Practical 1: Amplitude and Phase Response

Objectives and Background

The amplifier that you will use in this Assignment is one that requires an input current signal and produces an output voltage signal. This type of amplifier is sometimes known as a current input amplifier, but its correct name is a transresistance amplifier.

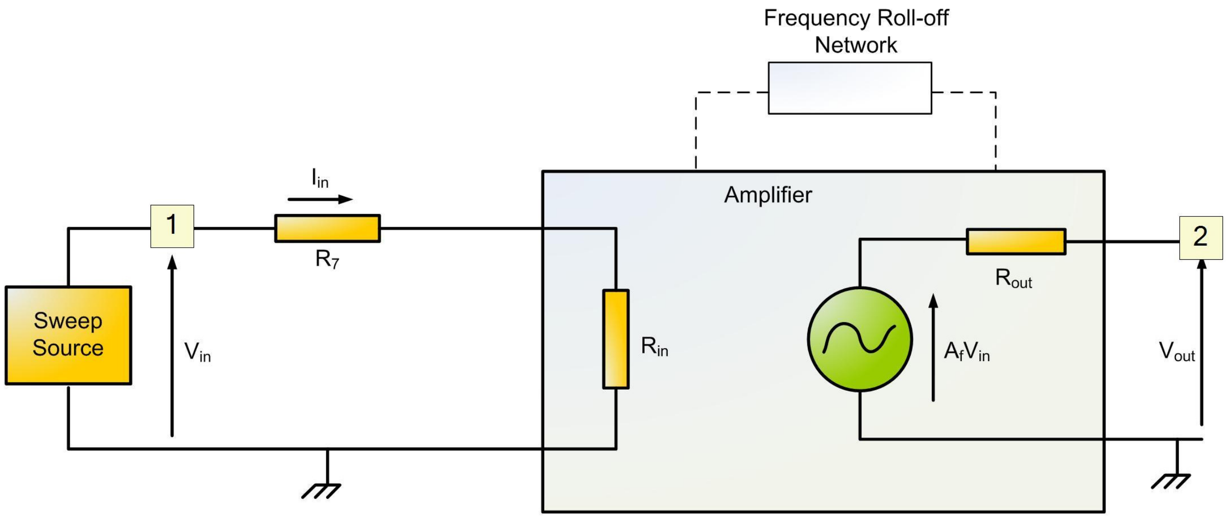


In this practical you will apply a sinusoidal voltage input to the Current Input Amplifier on the workboard and, initially, you will use the oscilloscope to see how the voltage gain of the amplifier varies with frequency.

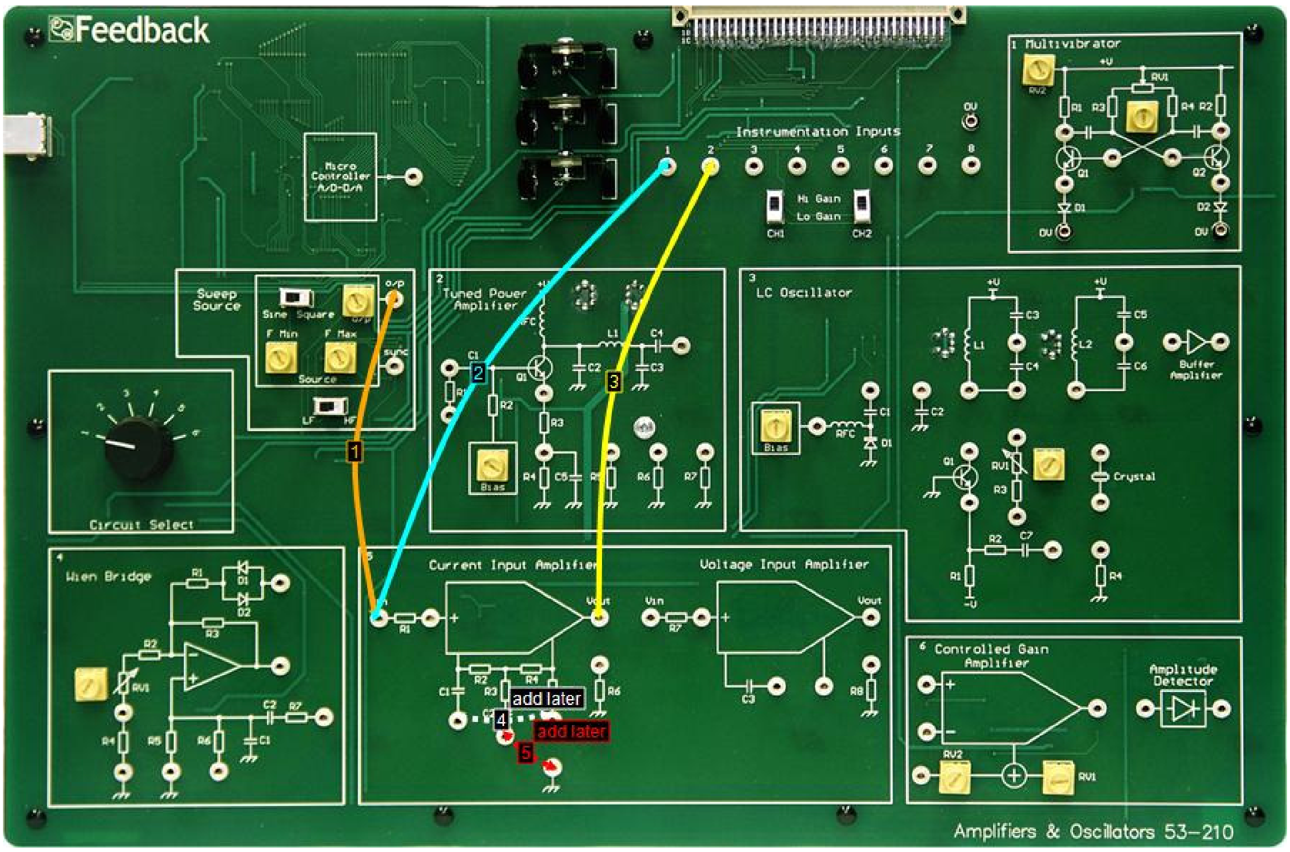
You will then use the Gain Phase Analyser test instrument (GPA) to plot the gain response automatically. You will also use the GPA to show how the phase shift through the amplifier varies with frequency.

You will see how the frequency response can be modified by the addition of a frequency roll-off network.

Block Diagram



Make Connections Diagram



Practical 1: Amplitude and Phase Response

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that you have the **Circuit Select** switch set to **5**.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard (directly below the Instrumentation Input sockets) and set both CH1 and CH2 to **Hi Gain**.

Identify the **Sweep Source** circuit block at the left-hand centre of the workboard. In this block, set the **Sine**/**Square** switch to Sine and the **LF**/**HF** switch (just below the Source block) to HF. Set both the **FMin** and **FMax** controls to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Open the oscilloscope and use the **o**/**p** control on the sweep source to set the its output amplitude (the yellow probe on monitor point 1) to approximately 0.8V pk-pk.

Open the frequency counter and monitor the frequency of the sweep source.

Use the cursors on the input and output traces of the oscilloscope to measure their positive and negative peak voltages. Calculate the peak-to-peak input and output voltages for the amplifier and thus determine the amplifier gain (as both a ratio and in dB).

Look at the input and output waveforms and estimate the phase shift (if any) between them.

Use the FMin control to vary frequency of the source and note how the input and output amplitudes change with frequency.

Use the FMin control to set the frequency to 2.5MHz.

As before, use the cursors on the input and output traces of the oscilloscope to measure their positive and negative peak voltages. Calculate the peak-to-peak input and output voltages for the amplifier and thus determine the amplifier gain (as both a ratio and in dB). Also, estimate the phase shift between them.

You should have found that the gain of the amplifier is lower at the high frequency and that, at the higher frequency, there is a phase shift between the input and the output waveforms, due to the delay through the amplifier.

Close all the test equipment and open the Gain Phase Analyser (GPA). The **Set Min Freq** button on the GPA is already selected by default. Set the Sweep Source FMin control to approximately 100kHz. Select the **Set Max Freq** button on the GPA and set the FMax control to approximately 8MHz.

Select the **Plot** button on the GPA to plot and calibrate the Bode (amplitude and phase) response of the amplifier between these frequencies. Note the variations with frequency.

Use the cursor to measure the gain and phase at 100kHz and at 2.5MHz. Compare these measurements with those achieved using the oscilloscope method. You may now appreciate the ease of use of the GPA compared with the oscilloscope for this purpose!

Now link between **C1** and **R5** as instructed by the Make Connections diagram (connection 4). This R5/C1 network should alter the high frequency response of the amplifier.

Use the GPA to give the Bode plot of the modified circuit. Note the plot and then recalibrate the GPA for a maximum frequency of approximately 2MHz.

Plot the response and compare the plots for the circuit with and without the R5/C1 network.

Remove the link between C1 and R5 (connection 4 on the Make Connections diagram) and connect **C2** to earth (connection 5 on the Make Connections diagram).

Plot the response and compare the plots for the circuit with and without the R3/C2 network.

Try to explain the results that you have found.

Practical 2: Measuring Input Resistance

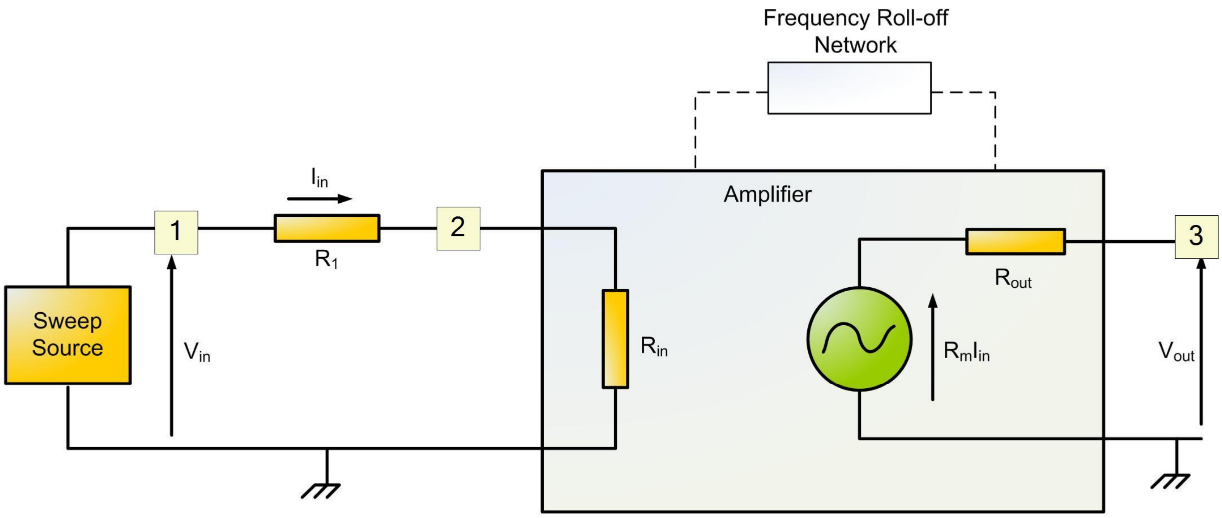
Objectives and Background

In this Practical you will apply a sinusoidal signal input to the amplifier and you will measure the amplitude of the voltage either side of a series input resistor, R1.

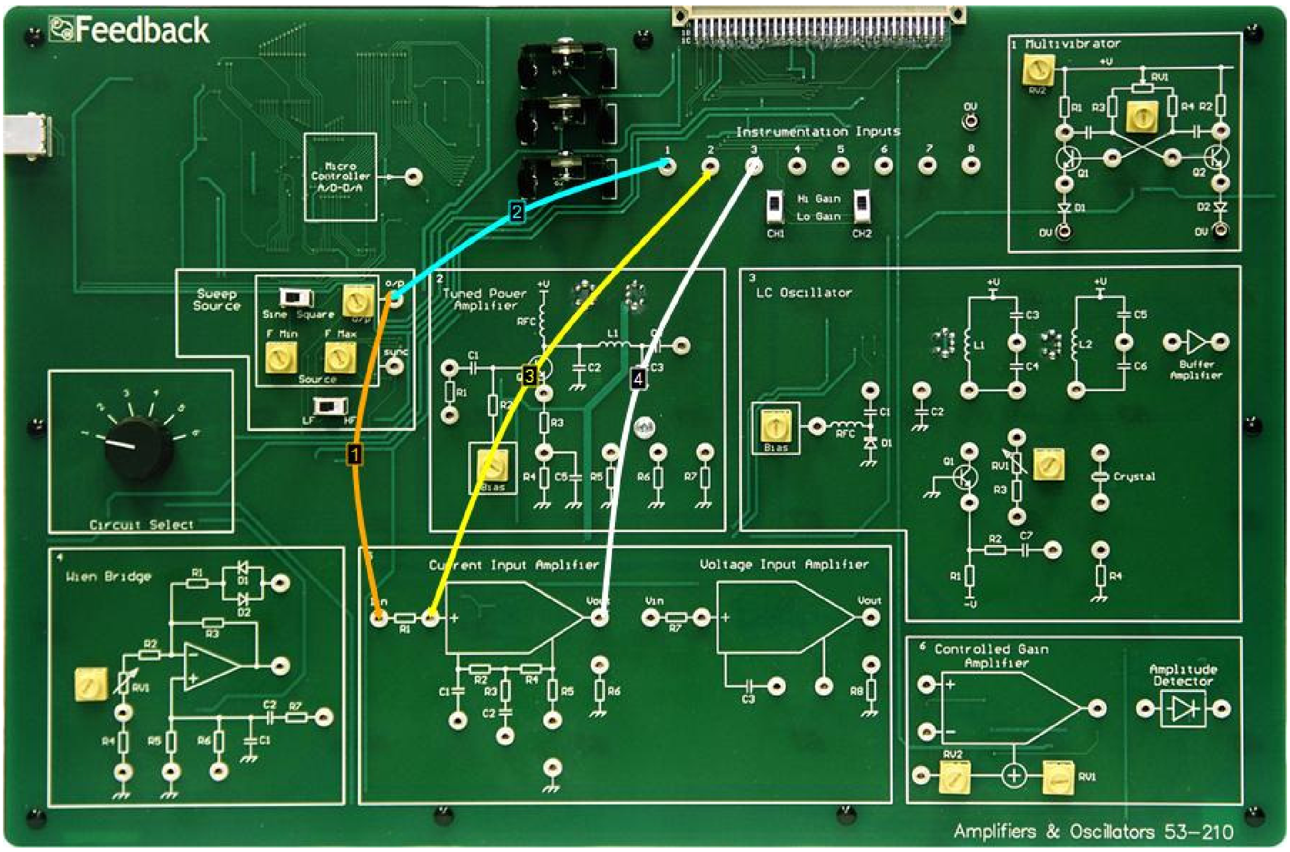
Knowing the value of R1, you will calculate the input current flowing and hence the input resistance of the amplifier.

You will perform these measurements at frequencies of 100kHz, 1MHz and 3MHz and compare the results.

Block Diagram



Make Connections Diagram



Practical 2: Measuring Input Resistance

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that you have the **Circuit Select** switch set to **5**.

Set both **CH1** and **CH2** to **Hi Gain**.

In the **Sweep Source** block, set the **LF/HF** switch to HF and the **Sine/Square** switch to Sine. Set both the **FMin** and **FMax** controls to their minimum (fully counter-clockwise) positions.

Open the oscilloscope and set the output amplitude from the Sweep Source (monitor point 1), using the **o/p** control, to be approximately 0.8V pk-pk.

Open the counter and use it to monitor the signal frequency.

Use the FMin control on the Sweep Source to set the frequency to approximately 100kHz.

Open the voltmeter and use it to measure the ac peak to peak amplitude of the input signal to the amplifier at both sides of R_1 (move the yellow probe between monitor points 1 and 2).

Calculate I_{in} and also R_{in} (knowing that $R_1 = 1k\Omega$).

You should find that the voltage at the amplifier end of R1 is very small. Because of this, accurate measurement of this voltage is difficult due to noise and pick-up associated with the other parts of the workboard and the measurement circuitry. Hence, the figures for the input current and resistance that you get will only be approximate. However, they do demonstrate the principles required.

Repeat the above procedure for input frequencies of approximately 1MHz and 3MHz.

Try to explain your results.

Practical 3: Measuring Output Resistance

Objectives and Background

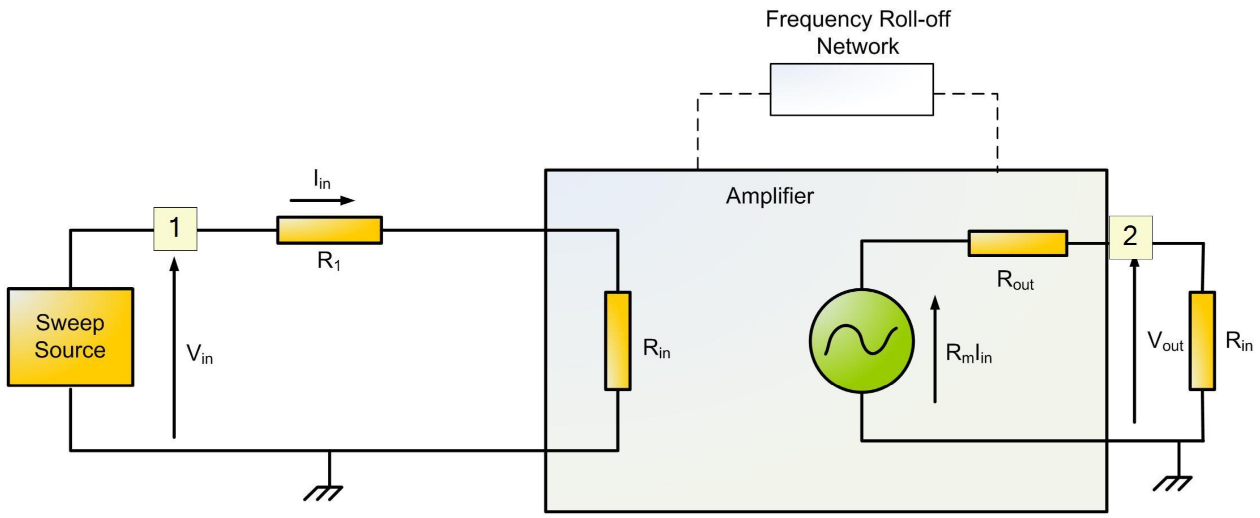
In this Practical you will determine the output resistance of the amplifier.

To achieve this, you will apply a sinusoidal signal input to the amplifier and you will measure the output signal amplitude, first with the output open circuit and then with a resistive load (R6) connected to the output.

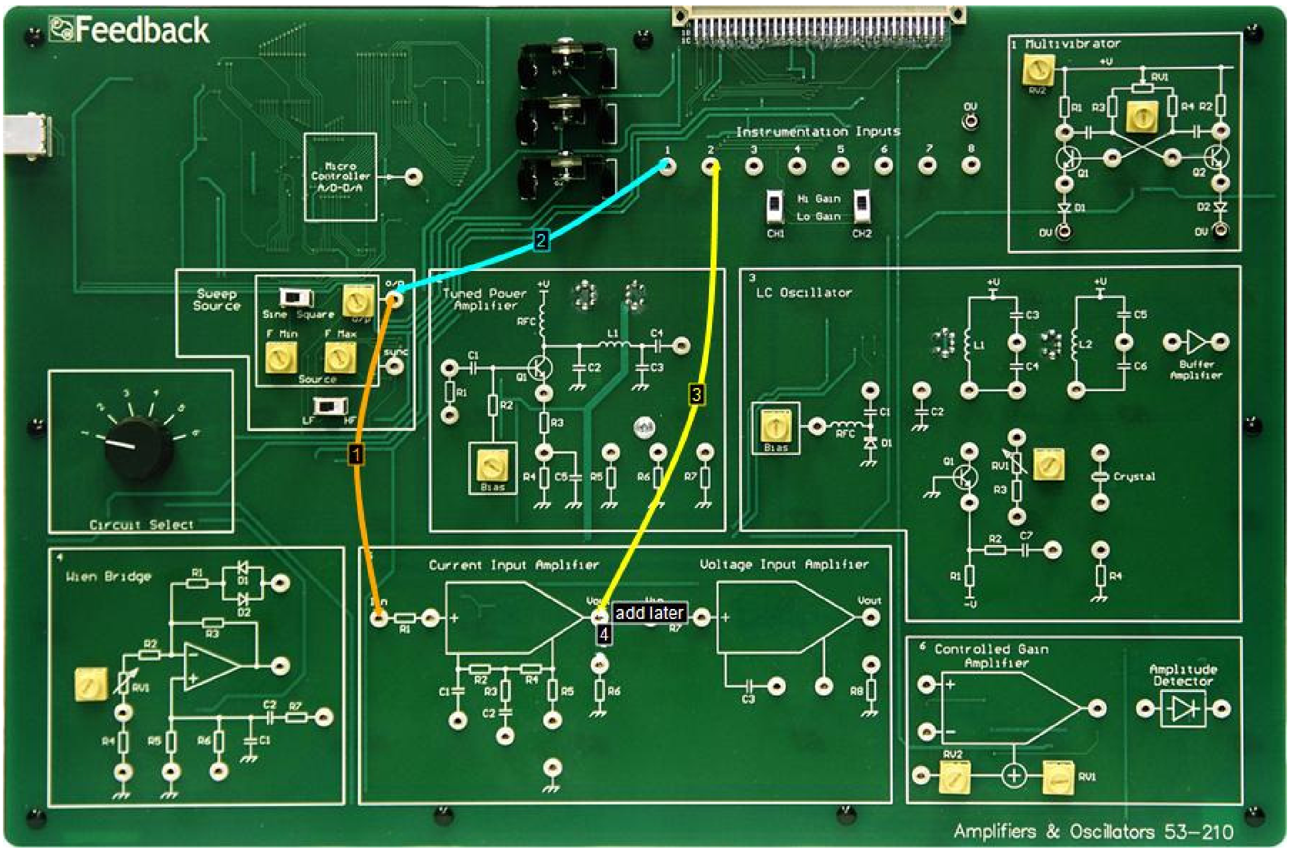
From these measurements, and knowing the value of R6, you will calculate the output resistance of the amplifier.

You will perform these measurements at frequencies of 100kHz, 1MHz and 3MHz and compare the results.

Block Diagram



Make Connections Diagram



Practical 3: Measuring Output Resistance

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that you have the **Circuit Select** switch set to **5**.

Set both **CH1** and **CH2** to **Hi Gain**.

In the **Sweep Source** block, set the to **LF**/**HF** switch to HF and the **Sine**/**Square** switch the Sine. Set both the **FMin** and **FMax** controls to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Open the oscilloscope and set the output amplitude from the Sweep Source (using the **o**/**p** control) to be approximately 0.8V pk-pk.

Open the frequency counter and use it to monitor the signal frequency.

Open the voltmeter and use it to measure the ac peak to peak amplitude of the output signal from the amplifier (the blue probe on monitor point 2).

Now connect the load resistor (R_6), as shown by connection 4 in the Make Connections diagram, and measure the new output voltage.

Calculate R_{out} (knowing that $R_6 = 220\Omega$).

Repeat the above procedure for input frequencies of approximately 1MHz and 3MHz.

Try to explain your results.

Controlled Gain Amplifier

Objectives

To appreciate the concept of a Controlled Gain Amplifier

To investigate the gain control characteristic and the phase shift associated with such an amplifier

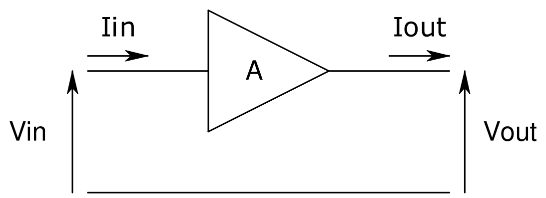
To show how such an amplifier can be used for automatic gain control (AGC)

To determine the characteristics of such an AGC system

Amplifiers

Gain

The block diagram of a general amplifier is given below.



The gain of the amplifier is denoted by the symbol A. The definition of A is given by:

$$\text{Gain, A} = \frac{(output\ power)}{(input\ power)}$$

This is called the **power gain** of the amplifier.

If a circuit does not have power gain, then it is not an amplifier!

As you can see, there is present input current and input voltage and, at the output, the corresponding output current and voltage. As power is the product of current and voltage, this gives the expression for the gain:

$$A = \frac{Vout.Iout}{Vin.Iin}$$

Now:

$$\frac{Vout}{Vin} = A_V \text{, the } \textbf{voltage gain} \text{ of the amplifier}$$

and:

$$\frac{Iout}{Iin} = A_I \text{, the } \textbf{current gain} \text{ of the amplifier}$$

Therefore:

$$A = A_V \, . \, A_I$$

Note: that it is quite possible for an amplifier to have a voltage gain of less than one;

however, its corresponding current gain must be high enough to give a power gain greater than unity for it to be classed as an amplifier. This also works the other way: if the current gain is less than one, the voltage gain must be high enough.

Going back to the expression for gain of:

$$A = \frac{Vout.Iout}{Vin.Iin}$$

This can be further re-arranged by defining two more relationships:

$\frac{Vout}{Iin} = G_m$, the **transresistance gain** of the amplifier

and:

$\frac{Iout}{Vin} = R_m$, the **transconductance gain** of the amplifier

Giving, therefore:

$$A = G_m \,.\, R_m$$

Using a similar argument to before, to qualify as an amplifier, it is quite possible for either one or the other of these terms to be less than one providing the product of the two is greater than unity.

Types of Amplifier

The input signal to an amplifier may be a current or it may be a voltage. Therefore, this gives rise to two types of amplifier: the current input amplifier and the voltage input amplifier.

Each of these types of amplifier may be further sub-divided, as each can give a current output or a voltage output.

There are, therefore, four general forms of amplifier.

Different applications require amplifiers to have different properties of amplification.

An example

Consider a fibre optic communications system with a great distance between transmitter and receiver. Because of the attenuation of light along the length of fibre it is often necessary to compensate for this loss by having ‘repeater’ circuits at intervals along the fibre. The purpose of these circuits is to detect the incoming light signal, convert it into an electrical signal that can be amplified and then used to drive a secondary light source that provides a regenerated signal for transmission further down the cable.

A typical sensor for the input of such a repeater is a photodiode. This will produce a current that is proportional to the light intensity. A typical output device might be a light-emitting diode, or perhaps a laser diode, which also needs a current signal to drive it. The amplifier within the repeater must thus take the small current output from the photodiode and amplify it to drive the LED or laser.

Therefore, the amplifier required has to be a **current amplifier**. The requirements for such a circuit are:

Its input should affect the signal current from the photodiode as little as possible,

It should have output circuitry that maximises the current transfer out of the amplifier,

It should have current gain.

To achieve this, it should have **as low an input resistance as possible** and its output should look like an ideal current source (i.e. it should have **as high an output resistance as possible**).

Another example

Consider a communications receiver system. Within the receiver there are circuits that perform functions such as high frequency amplification of the signals, the selection of the required signal (filters) and frequency translation of the signal to a lower frequency (mixers). Because of current limitations on the operating speed of analogue-to-digital converters and other digital circuitry, these functions are normally performed by analogue circuits. However, it is common practice to convert high frequency signals to a much lower frequency (often a few tens of kHz) so that such functions as demodulation or decoding and the final signal processing can be done using digital techniques. DSP (digital signal processing) circuits are widely used to do this.

Typically, DSP chips require input voltage signals of a few volts amplitude. The output signal from the analogue part of the receiver system may be only tens of millivolts in amplitude. An amplifier is therefore required to ‘bridge this gap’.

Therefore, this amplifier has to be a **voltage amplifier**. The requirements for such a circuit are:

Its input should affect the signal voltage from the analogue circuitry as little as possible,

It should have output circuitry that maximises the voltage transfer out of the amplifier,

It should have voltage gain.

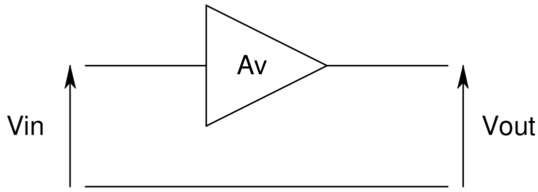
To achieve this, it should have **as high an input resistance as possible** and its output should look like an ideal voltage source (i.e. it should have **as low an output resistance as possible**).

You can imagine that the requirements for the two example amplifiers above will result in completely different circuitry to satisfy them.

The two other types of amplifier also find uses in electronics and communications systems. For example, a field effect transistor amplifier is an example of a **transconductance amplifier** and **transresistance amplifiers** are widely used in audio mixers and digital-to-analogue converters.

The Voltage Amplifier

A **voltage amplifier** is one to which an input voltage is applied and an output voltage results. The block diagram for such a system is given below.



The **voltage gain** of such an amplifier is given by

$$\text{Voltage gain,} \quad A_v = \frac{V_{out}}{V_{in}}$$

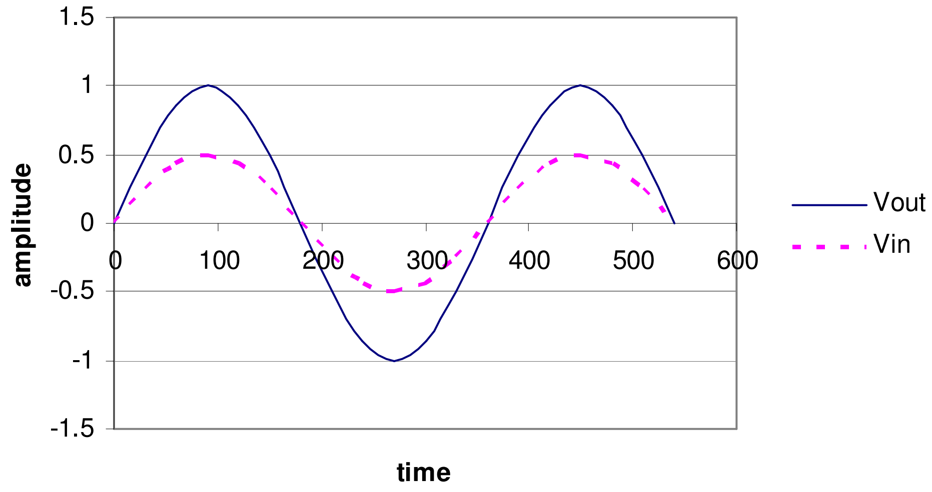
Ideally, connecting a voltage amplifier into a circuit should not affect the input voltage in any way. To achieve this, the input impedance of the amplifier should be as high as possible.

Also, ideally, connecting a load onto the output of the amplifier should not change the output voltage in any way. For this to happen, the output impedance of the amplifier should be as low as possible.

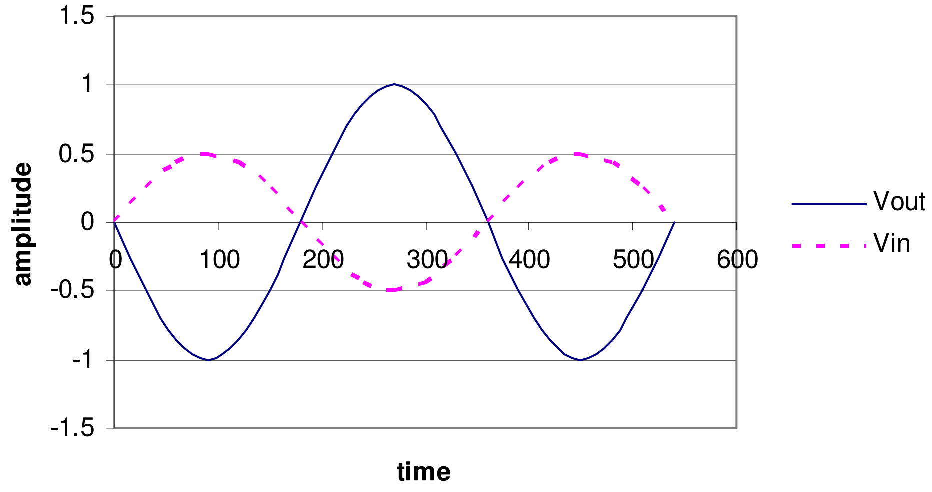
The ideal properties for a voltage amplifier are thus:

Voltage gain	Yes
Input impedance	Very high
Output impedance	Very low

The output voltage may follow the input voltage directly, or it may be inverted in polarity. This is illustrated below:



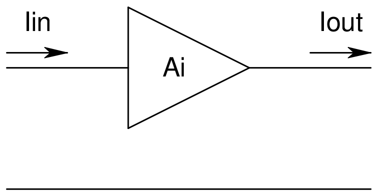
This shows the output following the input directly. The output is said to be **in phase** with the input. Another name for such an amplifier is a **non-inverting amplifier**. Note, also, that the output is amplified with respect to the input.



This shows the output inverted with respect to the input. The output is said to be in **anti-phase** with the input. Another name for such an amplifier is an **inverting amplifier**. Note, also, that the output is amplified with respect to the input.

The Current Amplifier

A **current amplifier** is one to which an input current is applied and an output current results. The block diagram for such a system is given below.



The **current gain** of such an amplifier is given by

$$\text{Current gain,} \quad A_i = \frac{I_{out}}{I_{in}}$$

Ideally, connecting a current amplifier into a circuit should not affect the input current in any way. To achieve this, the input impedance of the amplifier should be as low as possible.

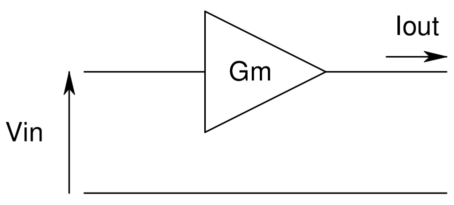
Also, ideally, connecting a load onto the output of the amplifier should not change the output current in any way. For this to happen, the output impedance of the amplifier should be as high as possible.

The ideal properties for a voltage amplifier are thus:

Current gain	Yes
Input impedance	Very low
Output impedance	Very high

The Transconductance Amplifier

A **transconductance amplifier** is one to which an input voltage is applied and an output current results. The block diagram for such a system is given below.



The **transconductance gain** of such an amplifier is given by

$$\text{Transconductance gain,} \quad G_m = \frac{I_{out}}{V_{in}}$$

Ideally, connecting a voltage amplifier into a circuit should not affect the input voltage in any way. To achieve this, the input impedance of the amplifier should be as high as possible.

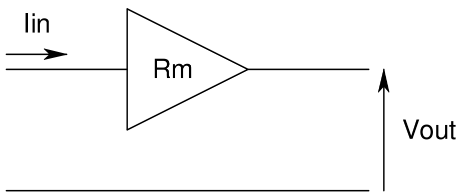
Also, ideally, connecting a load onto the output of the amplifier should not change the output current in any way. For this to happen, the output impedance of the amplifier should be as high as possible.

The ideal properties for a transconductance amplifier are thus:

Transconductance gain	Yes
Input impedance	Very high
Output impedance	Very high

The Transresistance Amplifier

A **transresistance amplifier** is one to which an input current is applied and an output voltage results. The block diagram for such a system is given below.



The **transresistance gain** of such an amplifier is given by

$$\text{Transresistance gain,} \quad R_m = \frac{V_{out}}{I_{in}}$$

Ideally, connecting a transresistance amplifier into a circuit should not affect the input current in any way. To achieve this, the input impedance of the amplifier should be as low as possible.

Also, ideally, connecting a load onto the output of the amplifier should not change the output voltage in any way. For this to happen, the output impedance of the amplifier should be as low as possible.

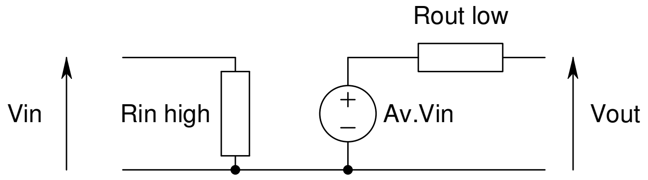
The ideal properties for a transresistance amplifier are thus:

Transresistance gain	Yes
Input impedance	Very low
Output impedance	Very low

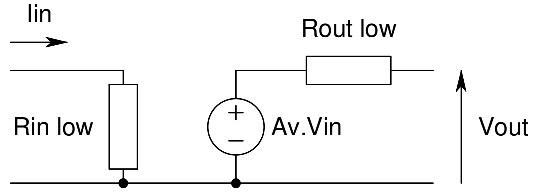
Summary of Amplifier Properties

A summary of the properties of the four types of amplifier, together with an equivalent circuit for each, is given below.

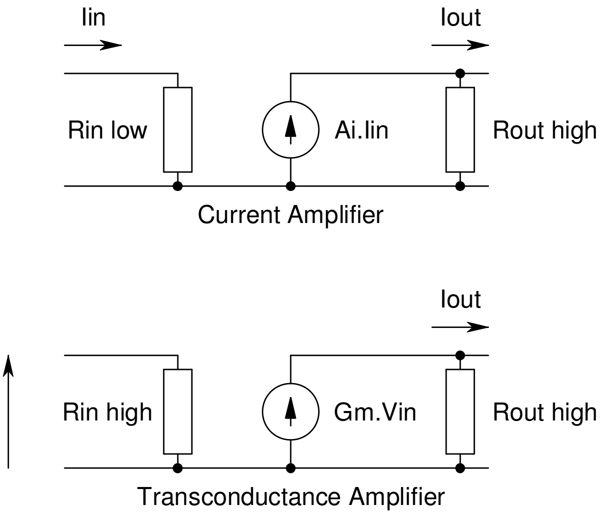
Amplifier Type	Gain Required	Input Resistance	Output Resistance
Voltage Amplifier	Voltage A_V	Very high	Very low
Current Amplifier	Current A_I	Very low	Very high
Transconductance Amplifier	Transconductance G_m	Very high	Very high
Transresistance Amplifier	Transresistance R_m	Very low	Very low



Voltage Amplifier



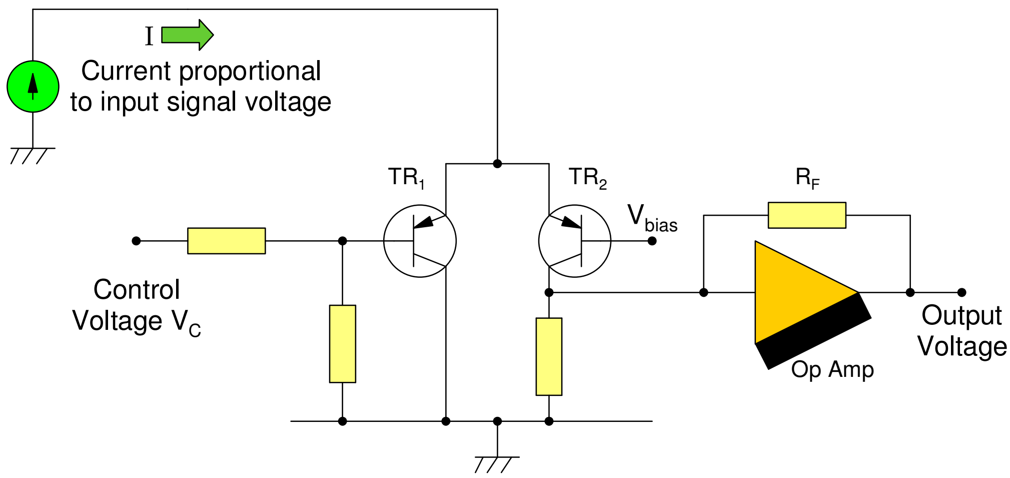
Transresistance Amplifier


Iin
Iout
Rin low
Ai.Iin
Rout high
Current Amplifier
Iout
Rin high
Gm.Vin
Rout high
Transconductance Amplifier

Voltage Control of Amplifier Gain

To be able to control the gain of an amplifier using an externally applied voltage is a useful property, as there are many situations in communications work where signals may vary in strength, so that the amount of amplification, i.e., the gain, needed to produce the desired signal strength also varies.

The Controlled Gain Amplifier on the workboard uses the properties of a differential amplifier to produce gain control. A simplified circuit of the amplifier is shown below.



The input signal voltage to be amplified is first converted to a proportional current (I) that is then applied to a differential pair of transistors (TR_1 and TR_2). The base of TR_1 is connected to the Control Voltage input whilst TR_2 base is connected to a fixed bias voltage.

The current flowing through TR_2 is converted to an output voltage by R_F and the Op Amp.

By changing the fraction of the signal current "I" that flows through TR_2, the gain is changed. This is done by changing the voltage applied differentially to the bases of TR_1 and TR_2.

For example, with the control voltage V_C = 0V, TR_1 conducts heavily and TR_2 is off. With none of "I" flowing through R_F, the circuit's input to output gain is strongly attenuated.

With $V_C >> V_{bias}$, TR_1 is off and the entire signal current flows through TR_2 to R_F, producing maximum gain.

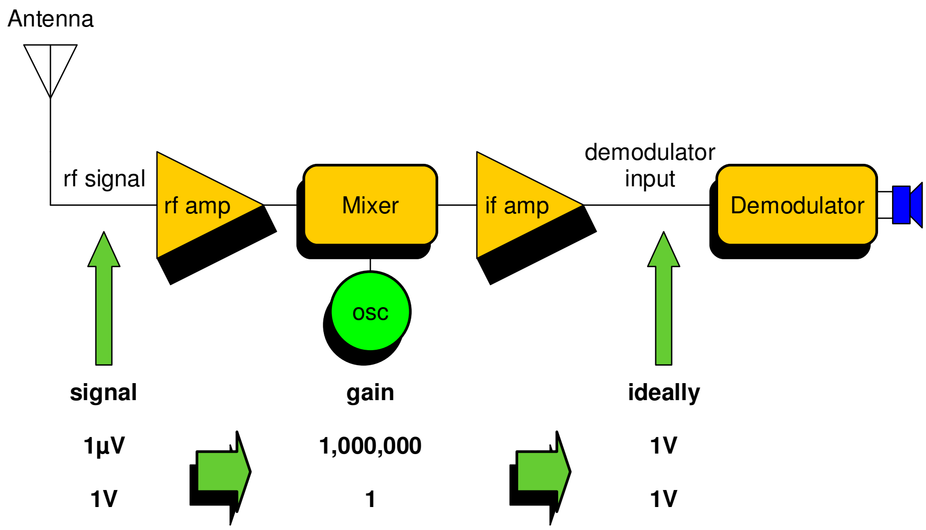
With V_C set to V_{bias} (the bias voltage on TR_2) the bases of TR_1 and TR_2 are set to the same voltage and thus they will have the same collector currents - equal to one half of the signal current "I". Thus the gain will be approximately one half of the maximum gain.

Any intermediate value of V_C will give a corresponding intermediate gain.

On the workboard, the control voltage can be applied from a potentiometer (RV1) or from an external source via a 2mm socket. The total control voltage is the sum of these two inputs. The control RV2 sets the gain of the summing amplifier.

Automatic Gain Control (AGC)

The amplitude of signals, especially RF signals, can vary over an enormous range. The signal from a distant transmitter arriving at an aerial (antenna) terminal of a radio receiver may be a microvolt or less. But if the transmitter is near-by that signal may well be several volts in amplitude.



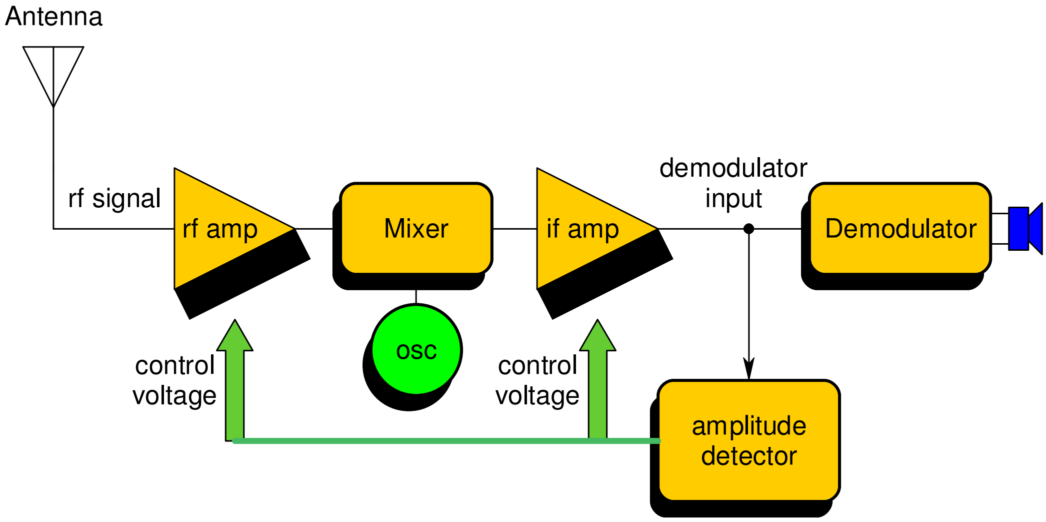
The signal required at the demodulator of a receiver is typically in the order of 0.2V to 10V. To achieve this, one or more amplifiers process the aerial signal on its way to the demodulator. To achieve, say, 1V at the demodulator, with input signals ranging from 1 microvolt to 1 V the gain has to be varied in the ratio one million to 1.

The control of the gain of an amplifier is therefore very important.

In fact, several amplifying stages in the receiver are normally gain-controlled to achieve this wide range.

The stage gains could be manually controlled; however, it is more usual to control them automatically. This is the process of AGC (automatic gain control). To do this, a feedback system is required that senses the signal voltage at the demodulator input and produces a control voltage proportional to this signal voltage. If the signal voltage falls (due to a fall in input RF signal) the system automatically adjusts the control voltage such that the gain of the system is increased to compensate. If the RF signal voltage rises, the control voltage produce is such that the system gain is reduced.

An ideal AGC system would produce a constant signal amplitude at the demodulator for a very wide range in RF input signal amplitudes.



The above diagram shows a typical receiver AGC system.

Practical 1: Controlled Gain Amplifier

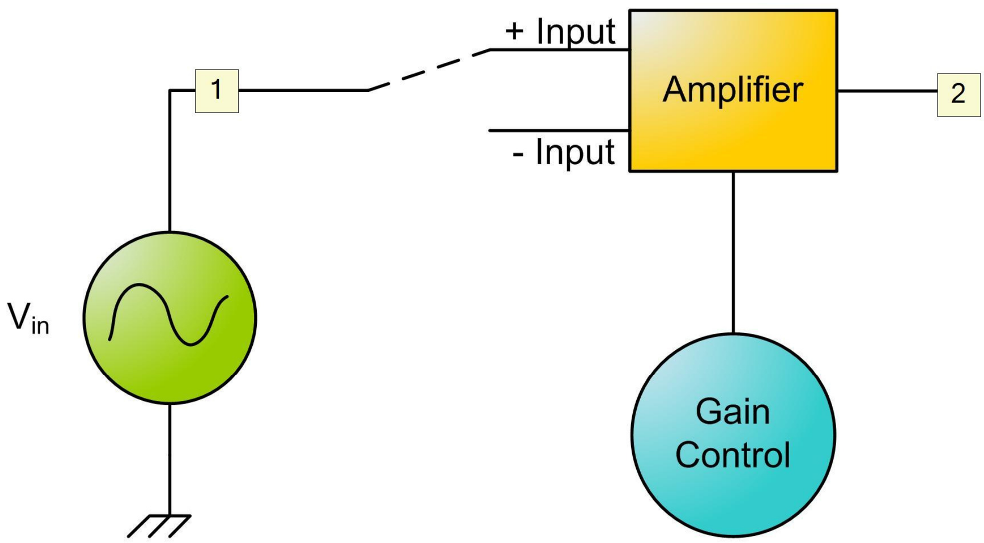
Objectives and Background

In this Practical you will investigate the operation and characteristics of the Controlled Gain Amplifier provided on the Workboard. You will apply an input signal from the Sweep Source and see how the output of the amplifier changes as the amplitude of the input is changed.

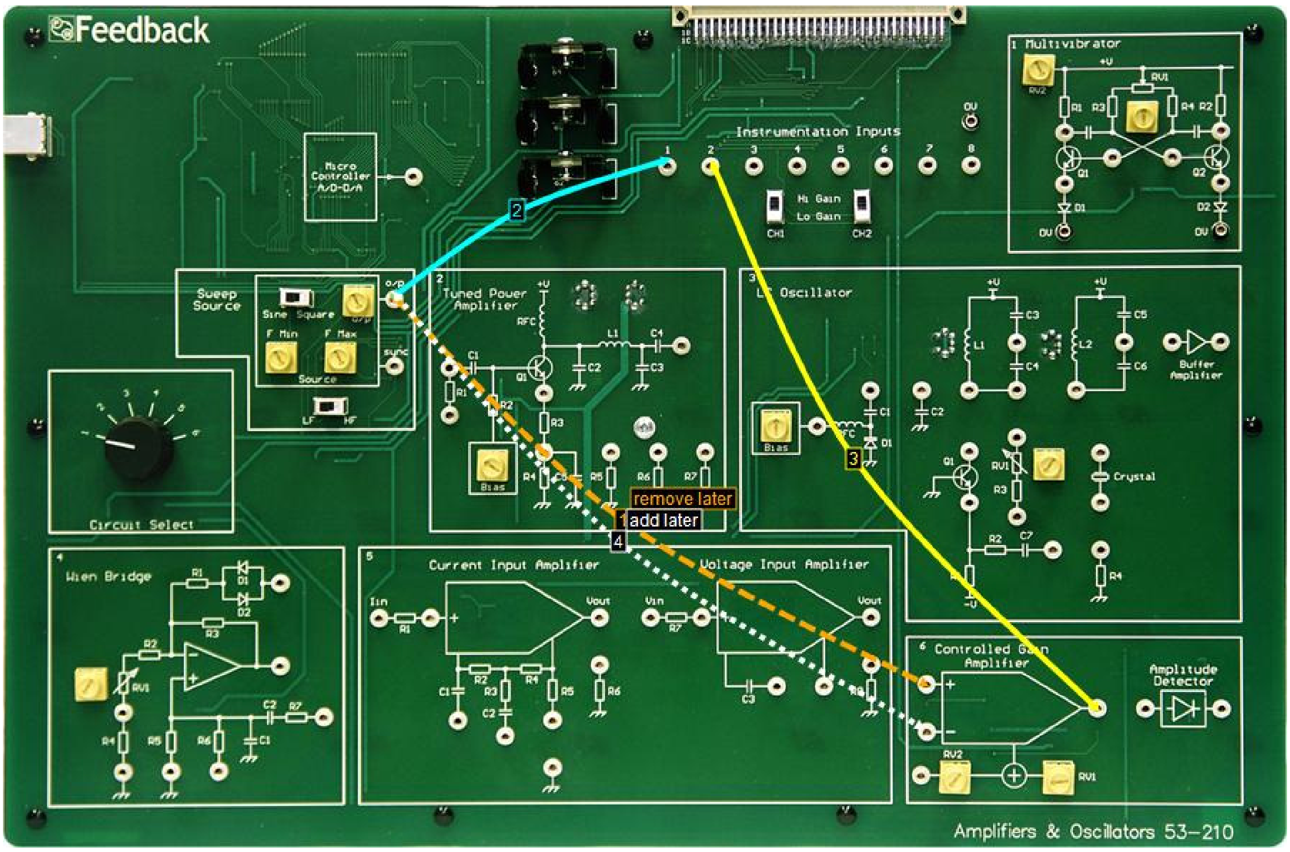
You will determine the phase relationship between the output and the input for the two input terminals of the amplifier.

You will vary the threshold of the gain control and see the effect.

Block Diagram



Make Connections Diagram



Practical 1: Controlled Gain Amplifier

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that you have the **Circuit Select** switch set to **6**.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set CH1 to **Lo Gain** and CH2 to **Hi Gain**.

Identify the **Sweep Source** circuit block at the left-hand centre of the workboard. In this block, set the **Sine**/**Square** switch to Sine and the **LF**/**HF** switch (just below the Source block) to HF. Set both the **FMin** and **FMax** controls to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Open the oscilloscope and use the **o**/**p** control on the sweep source to set the its output amplitude (the yellow probe on monitor point 1) to approximately 0.8V pk-pk.

Open the frequency counter and monitor the frequency of the sweep source.

Use the FMin control on the Sweep Source to set the input waveform frequency to approximately 1MHz.

Set the **RV1** control associated with the Controlled Gain Amplifier to its maximum (fully clockwise) position.

Look at the input and output waveforms and estimate the phase shift (if any) between them.

Vary the position of the RV1 control and observe any change in gain. Reset the control to maximum.

You should see that the gain of the amplifier can be changed by varying the voltage applied to the summing point (shown by the cross in the circle) by the potentiometer RV1.

Change the input to the amplifier to the – input, as shown by connection 4 in the Make Connections diagram and note any changes to the output of the amplifier.

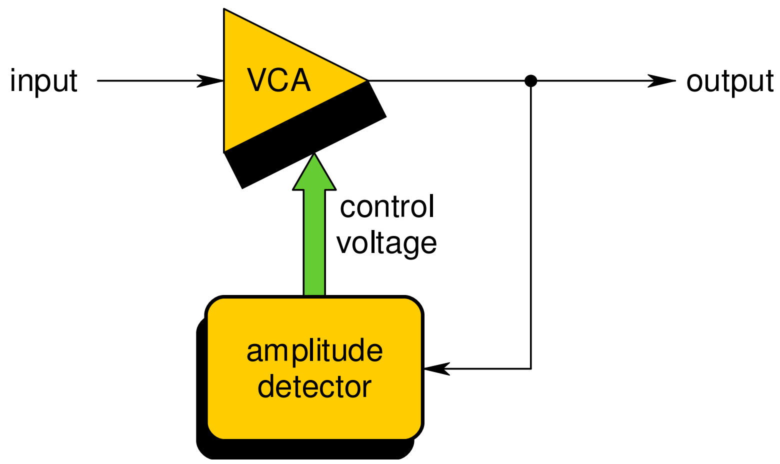
Again, vary the position of the RV1 control and observe any change in gain.

Practical 2: Automatic Gain Control

Objectives and Background

In this Practical you will connect the Controlled Gain Amplifier that you investigated in Practical 1 to an amplitude detector circuit and use the output from this to control the amplifier gain automatically.

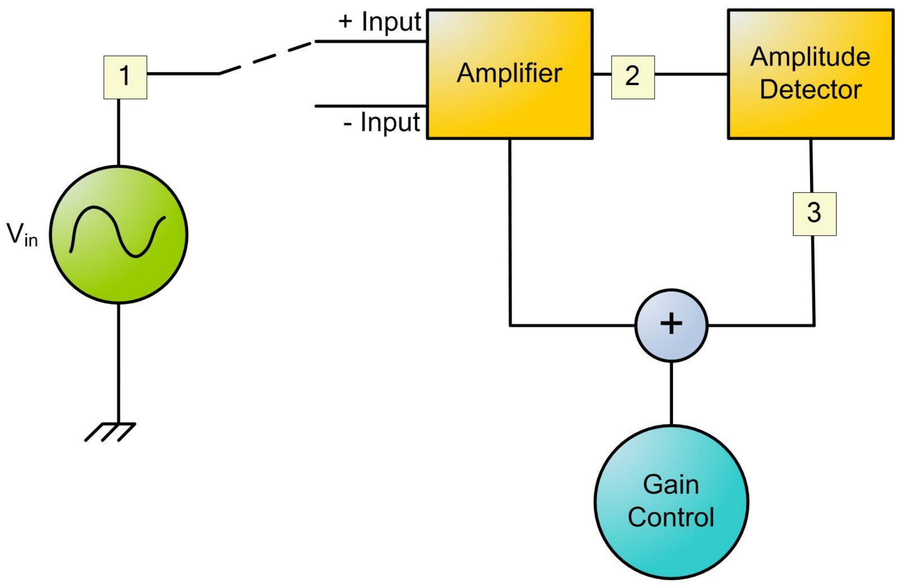
The block diagram of the system that you will use is shown below.



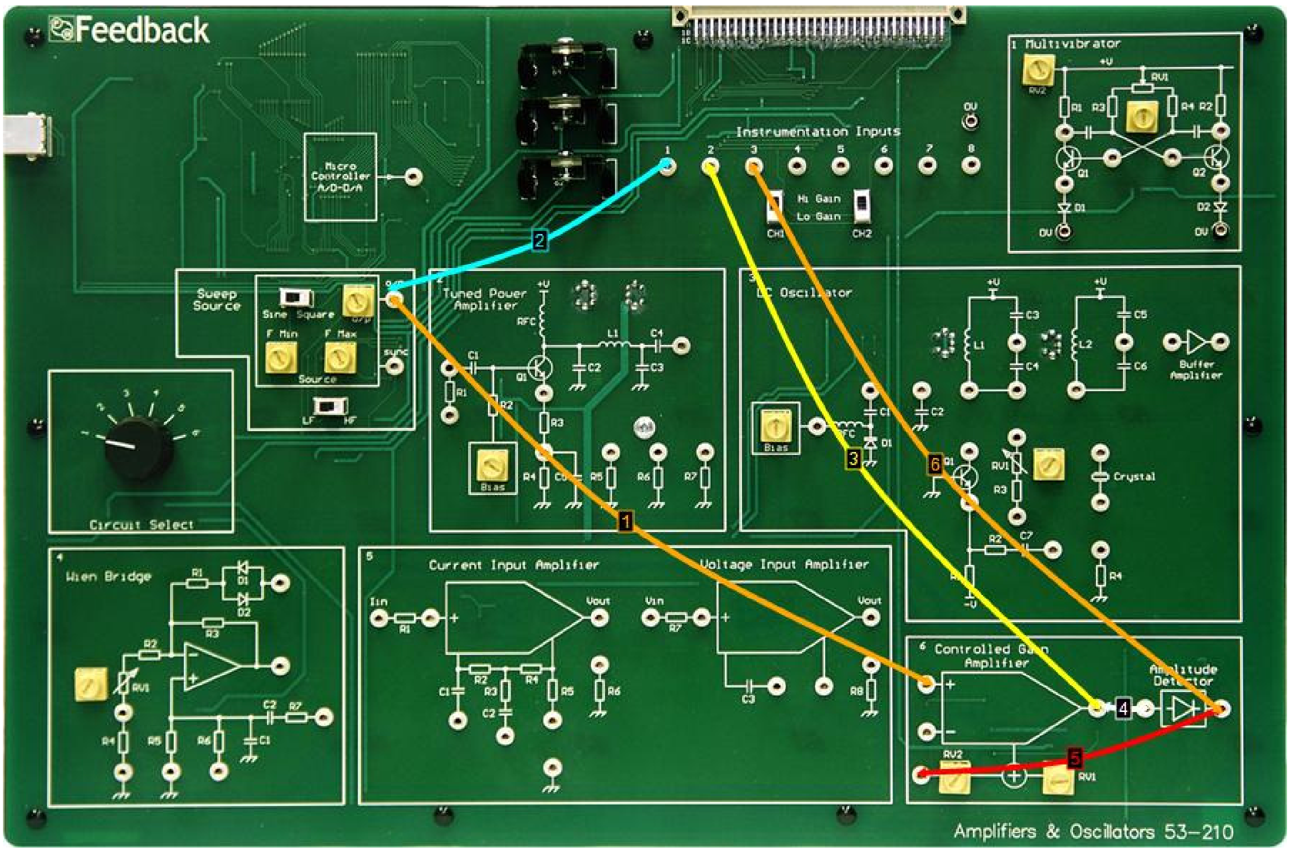
You will measure the input and output voltages of the amplifier for various settings of the two controls associated with the circuit and see how one sets the AGC threshold level and the other sets the gain of the control system.

You will plot graphs of these characteristics.

Block Diagram



Make Connections Diagram



Practical 2: Automatic Gain Control

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that you have the **Circuit Select** switch set to **6**.

Set **CH1** to **Lo Gain** and **CH2** to **Hi Gain**.

Set the **Sweep Source** to **HF** and **Sine**. Set both the **FMin** and **FMax** controls to their minimum (fully counter-clockwise) positions.

Open the counter and use it to monitor the signal frequency.

Use the FMin control on the Sweep Source to set the frequency to approximately 1MHz.

Firstly you will investigate the effect of the setting of **RV1** on the operation of the circuit. Set **RV2** to half scale and do not re-adjust it for this part of the Practical.

Open the oscilloscope and set the output amplitude from the Sweep Source (the yellow probe on monitor point 1), using the o/p control, to be zero. The blue trace (monitor point 2) shows the output of the amplifier. It should also be zero.

Open the voltmeter and use it to measure the dc voltage at the output of the Amplitude Detector (the orange probe on monitor point 3). This is the control voltage that is applied to the amplifier.

Set RV1 to half scale.

Increase the input voltage to the amplifier (using the o/p control on the Sweep Source) in steps of 0.1V pk-pk from zero up to 0.8V pk-pk and observe carefully how the output voltage of the amplifier and also the dc control voltage change.

Open the spreadsheet and enter your results in the table.

Repeat the above measurements for RV1 set to positions of approximately one quarter and then two thirds of full scale position. Do not re-adjust RV2 while you are taking the measurements.

Now you will investigate the effect of the setting of RV2 on the operation of the circuit. Set RV1 to half scale and do not re-adjust it for this part of the Practical.

Set RV2 to its maximum (fully clockwise) position. Take new measurements of V_{out} against V_{in} and record them in Chart 2 of your spreadsheet.

Repeat the above measurements for RV2 set to its minimum position (fully counter-clockwise). Do not re-adjust RV2 while you are taking the measurements.

You should now be in a position to see how the AGC circuit operates and the effects that the two controls have on its operation.

The LC Oscillator (1)

Objectives

To become familiar with the operation of a tuned amplifier

To appreciate the requirements for a circuit to oscillate and produce a sinusoidal output signal

To verify that the above requirements are met for the LC oscillator circuit

To investigate the factors that determine the amplitude of the oscillator output signal

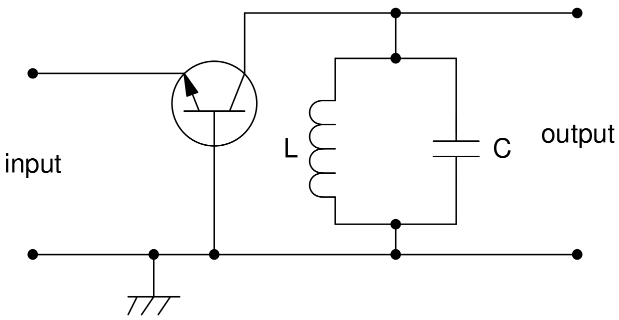
To investigate the factors that effect the amplitude and frequency stability of the oscillator

Tuned Amplifiers

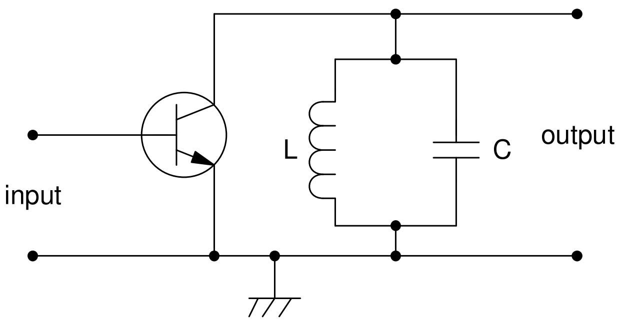
A tuned amplifier is one that has an LC tuned circuit, generally, as the load for the amplifying device.

For a bipolar transistor amplifier, the most common connections for the device used in a tuned amplifier are common emitter or common base. In each of these connections the output terminal of the transistor is the collector. The tuned circuit is thus most commonly connected in the collector circuit.

Simplified circuits, excluding any power supplies or components required to bias the transistor correctly, are shown below.



Common Base Connection



Common Emitter Connection

In both cases, above, the impedance of the parallel LC tuned circuit (ignoring any component losses and resistances) is given by:

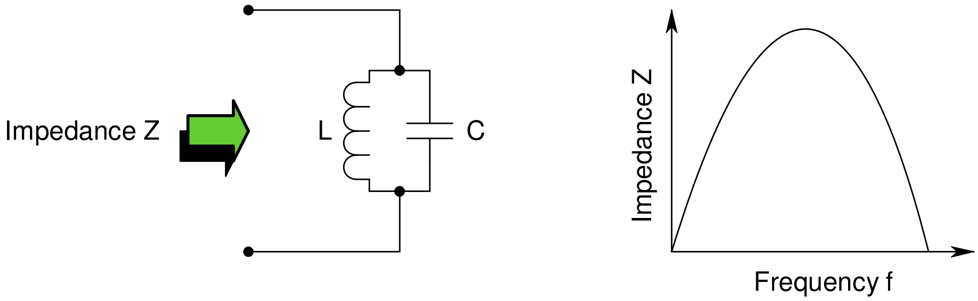
$$Z = \frac{j\omega L}{1 - \omega^2 LC}$$

This is frequency (ω) dependant. It will be a maximum when

$$\omega^2 LC = 1$$

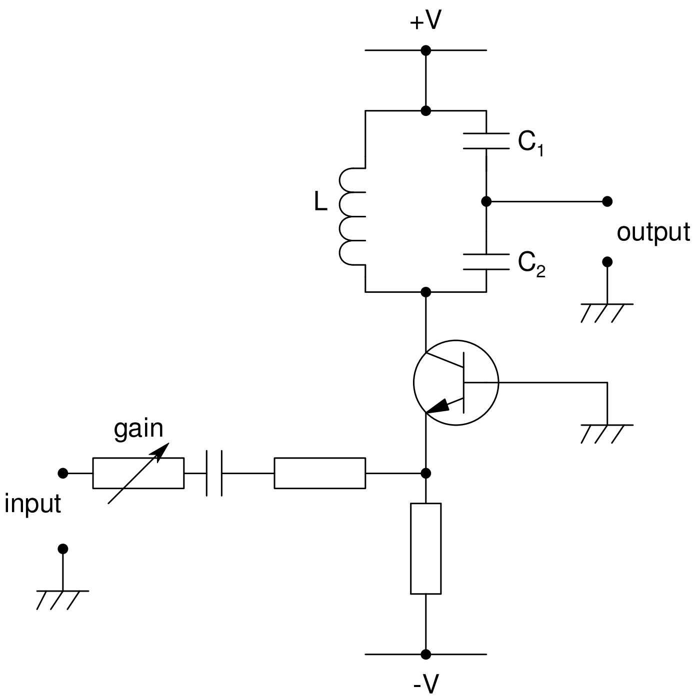
Thus the gain of the amplifiers will be a maximum at this frequency. This frequency is the **resonant frequency** of the tuned circuit.

At any frequency away from the resonant frequency the impedance of the tuned circuit will drop, as shown below.



Thus the frequency response of the amplifier will be of a similar form:

A typical common base circuit (that is used on the workboard), complete with bias resistors, capacitors, etc, is shown below.



The two capacitors in the tuned circuit form, effectively, a capacitive potential divider. This is required for the purposes of the assignment. The equivalent value (C) of these two capacitors is given by their series combination.

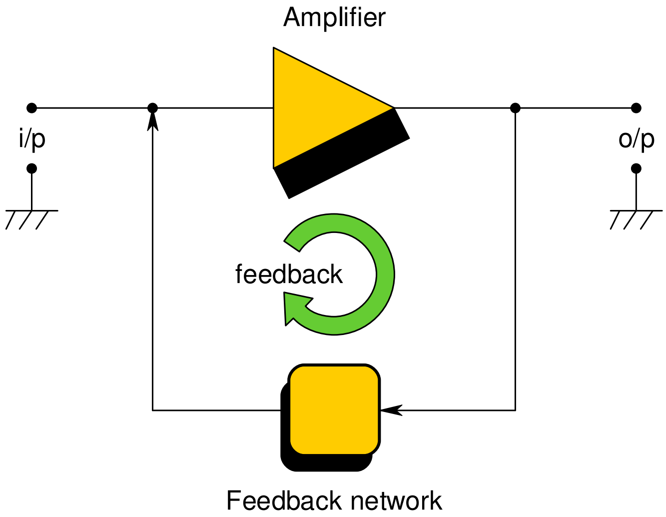
$$C = \frac{C_1 C_2}{C_1 + C_2}$$

Requirements for Oscillation

An oscillator is a circuit that produces an output signal for no input signal.

An alternative way of thinking of an oscillator is that it is a circuit that converts dc power (which comes from the dc supply to the circuit) into signal power that is available at the output terminals.

Consider an amplifier with feedback applied from the output back to the input, as shown in the following diagram.



The signal fed back can be such that it either helps to cancel or reinforce the original input signal, depending on the **phase** of the feedback signal.

Signals that are fed back in phase with the original input signal will reinforce its effect

Signals that are fed back in anti-phase with the original input signal will reduce its effect

This gives rise to two forms of feedback:

Signals that are fed back in phase with the original input signal give **positive feedback**

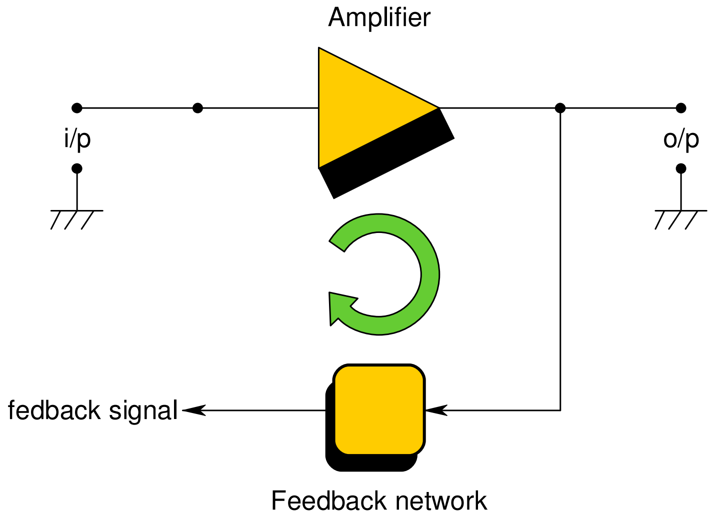
Signals that are fed back in anti-phase with the original input signal give **negative feedback**

An oscillator requires positive feedback to be applied.

The combination of the amplifier and the feedback network is known as the **feedback loop**. For positive feedback to be applied the signal fed back round the loop must be such that it is **in phase** with the original input signal. This means that the **total** phase shift, first through the amplifier and then through the feedback network, must be 0 degrees. This is usually referred to as **the loop phase shift**.

Having determined that loop phase shift must be zero to get full signal reinforcement, the next question is “how much signal needs to be fed back to get sustained oscillation?”

To answer this question consider the circuit above, but with the feedback loop ‘broken’, as shown in the diagram below.



Suppose the amplifier had a gain of 3 and the feedback network attenuated the signal by a factor of 5 (i.e. it has a ‘gain’ of 0.2). This means that the feedback signal will be 0.2 of the amplified signal, i.e. 0.2 x 3 times the original input signal: = 0.6 of the input amplitude.

The combination of the amplifier gain and the feedback network attenuation is called the **loop gain**.

To be able to maintain an output, the attenuation of the feedback network must at least be made up by the gain of the amplifier. That **means that the loop gain must be at least unity**.

In this instance, the feedback signal is less than the original input signal so, if the feedback loop were reconnected and the input signal removed, there would not be enough feedback to maintain an output. The loop gain is less than unity. It is usually much easier to change the gain of an amplifier than the attenuation of the feedback network thus, to get unity loop gain, in this instance the amplifier gain must be increased to at least 5.

Thus, the criteria for oscillation are:

the loop phase shift must be zero and

the loop gain must be at least one

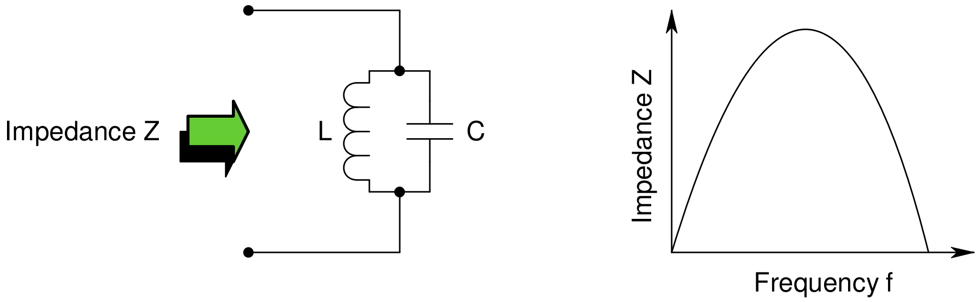
Practical 1: The Tuned Amplifier

Objectives and Background

The gain of an amplifier is dependant on the load that is connected to the output of it. Generally, the gain is proportional in some way to the value of the load impedance.

With a resistive load, the load impedance is just the resistance value and this is (ideally) constant with frequency. With a load that comprises inductor(s) and capacitor(s) the impedance is more complicated and is not constant with frequency.

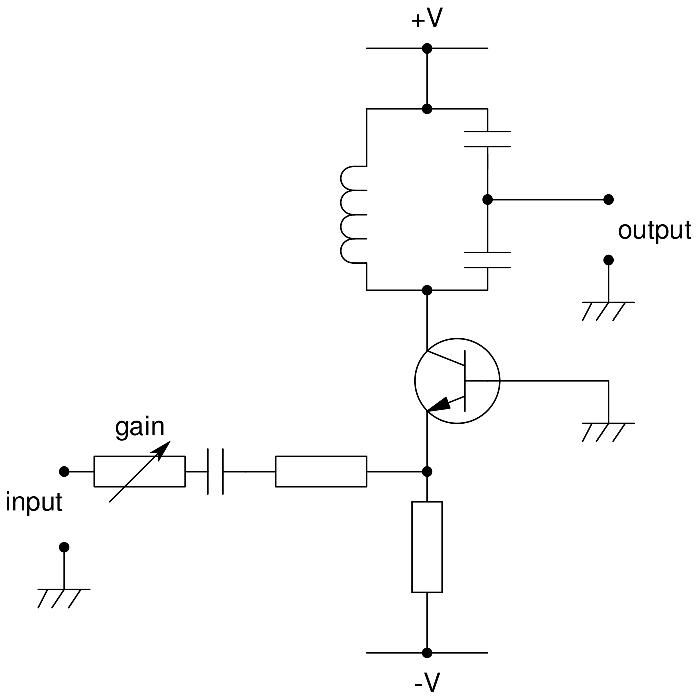
A parallel tuned (LC) circuit has a frequency selective response. A typical LC tuned circuit and its impedance response is given in the diagram below.



If such a parallel tuned circuit is connected as the load for a transistor, the gain of the transistor will be frequency dependant, with the maximum gain occurring at the frequency at which the load impedance is a maximum. This frequency will be at the **resonant** frequency of the tuned circuit.

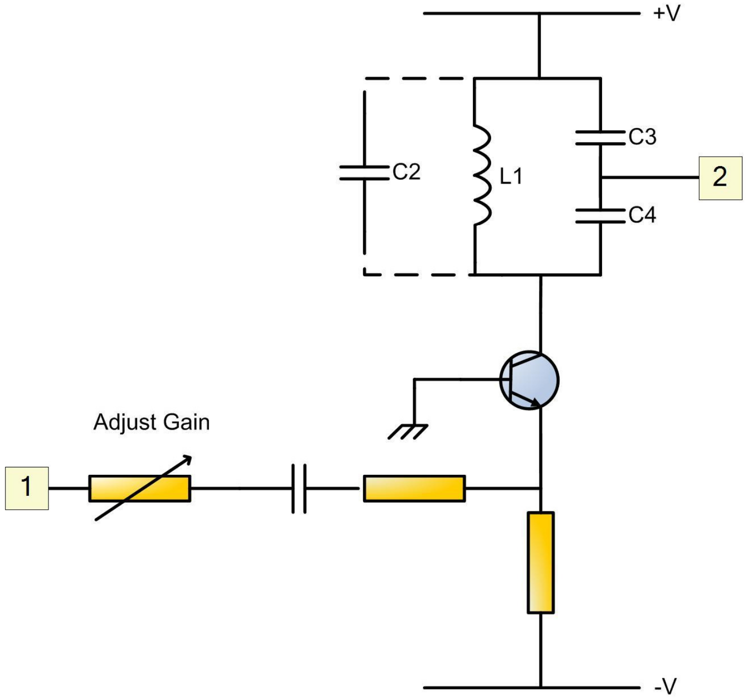
In this Practical you will connect a tuned circuit to the collector (output) of a transistor operating as a common base amplifier and you will investigate the frequency response of the amplifier using the oscilloscope, the frequency counter and the phasescope. As the circuit that you will use is a common base connection, the input is applied to the emitter of the transistor and the output is taken from the collector.

The circuit is given below.

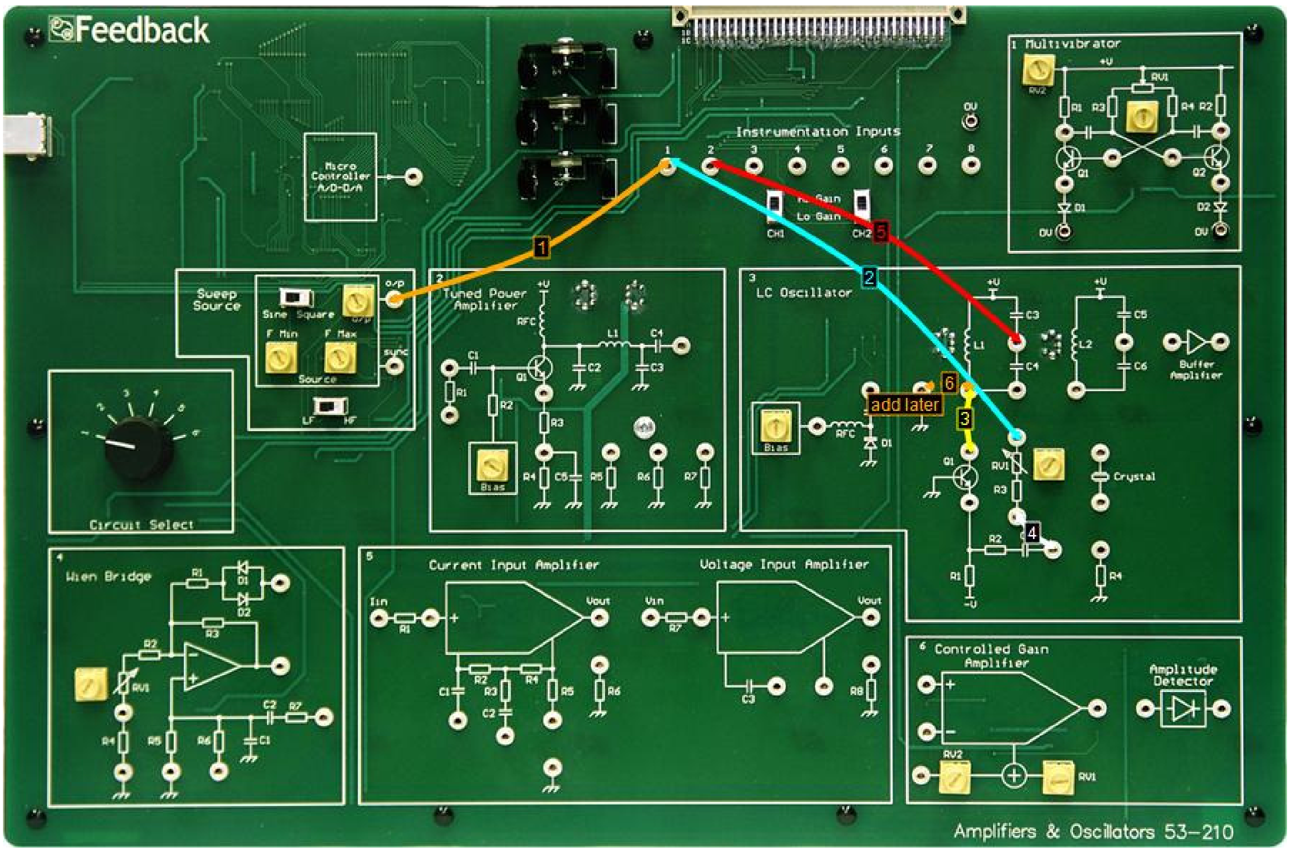


You will add capacitance to the tuned circuit and you will see the effect on the frequency of resonance.

Block Diagram



Make Connections Diagram



Practical 1: The Tuned Amplifier

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **LC Oscillator** circuit block located to the centre-right of the workboard.

Ensure that the **Circuit Select** switch, on the left of the workboard, is set to position **3**.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 switches to **Hi Gain**.

Identify the **Sweep Source** circuit block at the left-hand centre of the workboard. In this block, set the **Sine**/**Square** switch to Sine and the **LF**/**HF** switch (just below the Source block, to HF. Set both the **FMin** and **FMax** controls to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Set the **RV1** control in the LC Oscillator circuit to half scale.

Open the oscilloscope and use the **o**/**p** control on the Sweep Source to set its output to just under 1V pk-pk.

Open the frequency counter and monitor the frequency of the Sweep Source. Use the FMin control on the sweep source to vary the frequency of the source and see how the input and output amplitudes vary.

You should find that the input amplitude stays relatively constant but that the output amplitude peaks at one particular frequency. Note the frequency of this peak.

Carefully set the frequency to this peak. It should be very sharp.

Open the phasescope. Carefully adjust the source frequency about the peak and observe the phasescope. Use the frequency control to set the phase as close as possible to zero. This is the resonant frequency of the tuned amplifier. Note this frequency.

Also note that, around the resonant frequency, the phase changes much more rapidly than the amplitude.

Change the frequency from one well below resonance to one well above and note how the amplitude and phase change. Note the limits of phase.

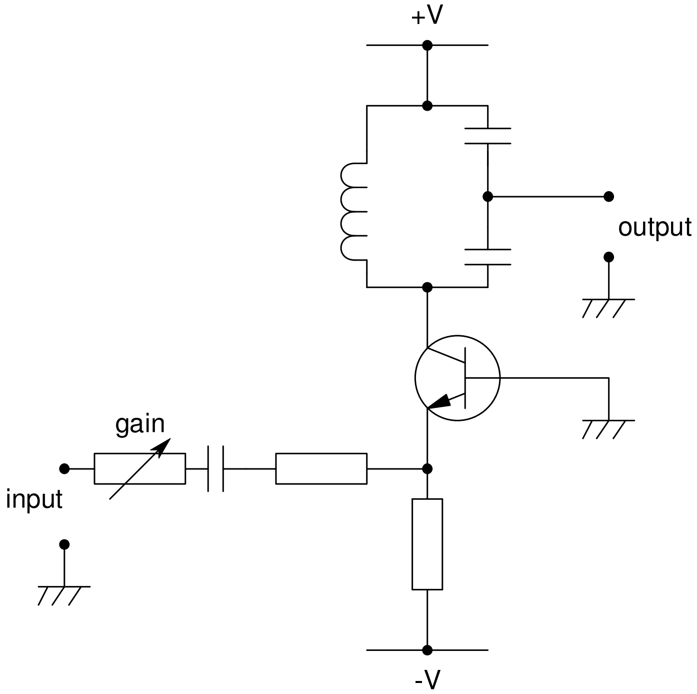
Now, connect **C2** across the tuned circuit by adding connection 6, as shown on the Make Connections diagram. Adjust the source frequency to find the new resonant frequency.

Practical 2: Using the Gain Phase Analyser

Objectives and Background

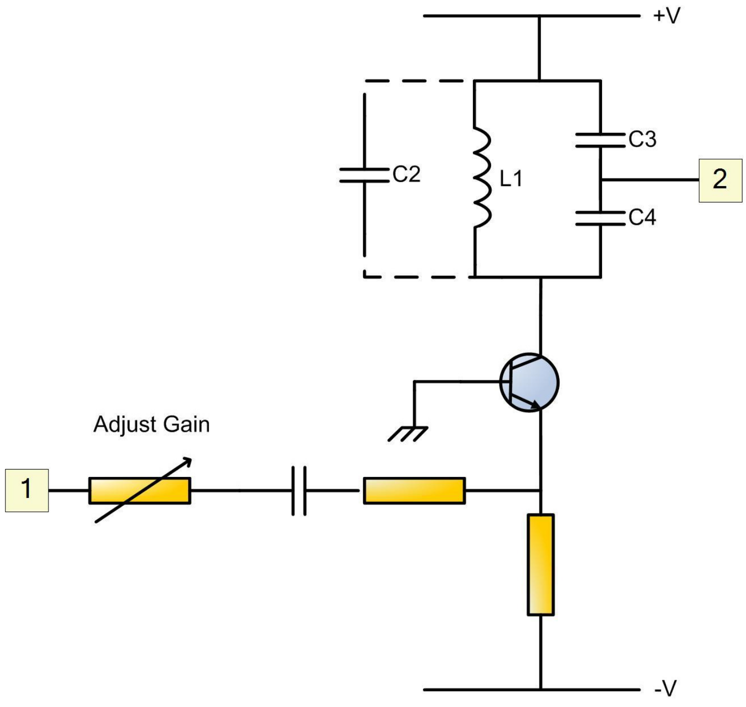
In this Practical you will use the Gain-Phase Analyser to investigate the same circuit that you used in Practical 1.

The circuit, as before, is given below.

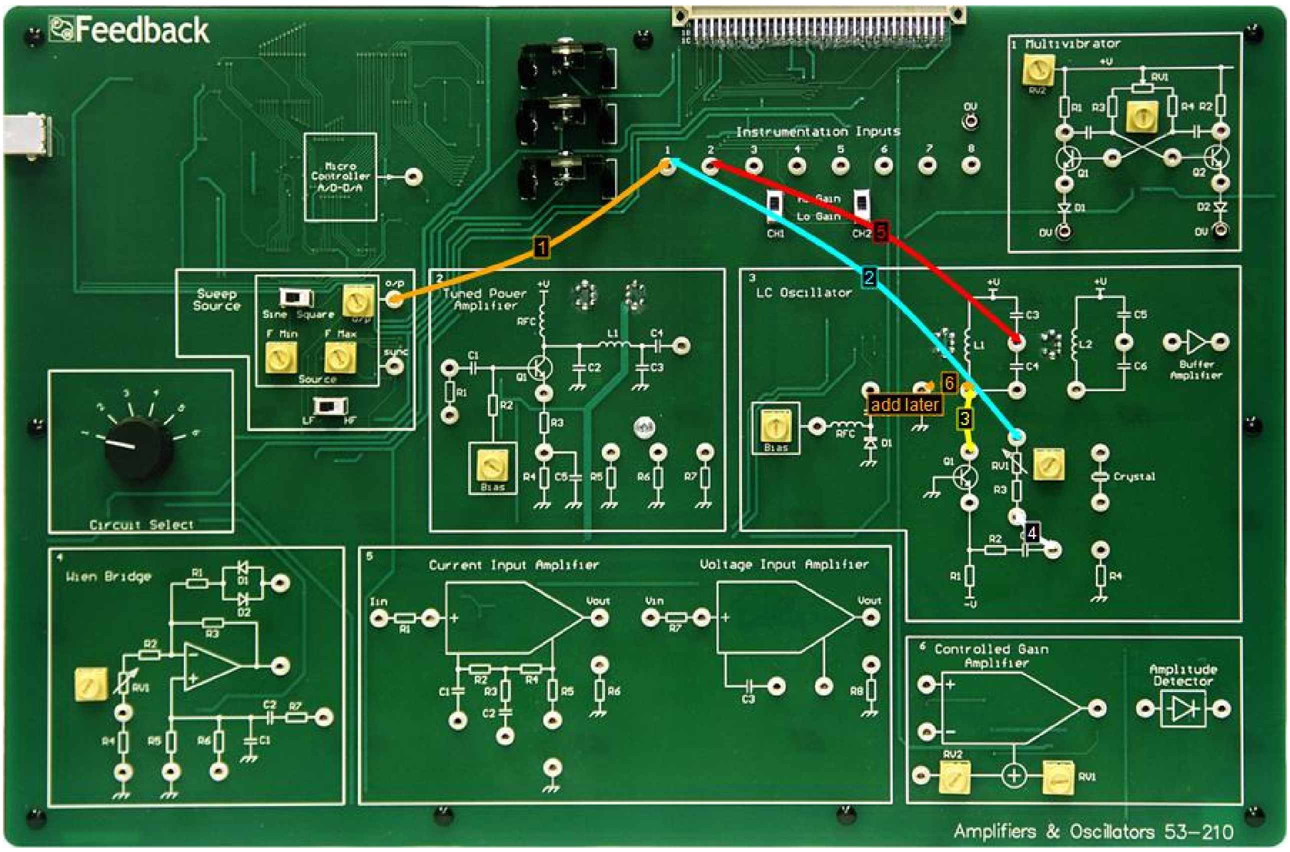


You will vary the gain of the amplifier using a potentiometer control and you will see the effect on the response.

Block Diagram



Make Connections Diagram



Practical 2: Using the Gain Phase Analyser

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware (the connections are the same as for Practical 1 however, ensure that you start with C2 **NOT** in circuit).

You can use the same initial control and switch settings as you had when you finished Practical 1.

Open the Gain Phase Analyser (GPA). By default, the **Set Min Freq** button is selected.

Use the **FMin** control of the Sweep Source to set the minimum sweep frequency to approximately 1.5MHz.

Click on the **Set Max Freq** button on the GPA. Use the **FMax** control of the Sweep Source to set the maximum sweep frequency to approximately 2.2MHz.

Now, click the **Plot** button on the GPA.

The GPA automatically gives you the Bode plot of the amplifier, showing the two traces: magnitude and phase.

Use your mouse to left click on the point where the phase trace passes through zero phase and read off the magnitude and frequency of this point.

Now vary the **RV1** control, in the **LC Oscillator** circuit block, and note the change in the displayed magnitude but little change in phase or resonant frequency.

Reset the RV1 control to approximately half scale.

Now, connect **C2** across the tuned circuit by adding connection 6, as shown on the Make Connections diagram. Use the cursor on the GPA display to measure the new resonant frequency.

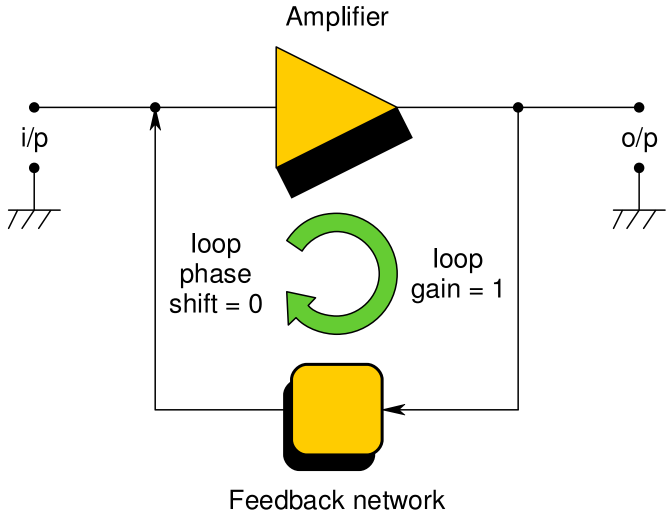
On the GPA, select a Nyquist plot by ticking the **Nyquist** box. Also tick the **Hi Res** box to get a better frequency resolution of the plot.

Use the slider bar at the left-hand side of the display to position the cursor on the zero phase line. Read off the frequency, magnitude and phase using the cursor.

Practical 3: Requirements for Oscillation

Objectives and Background

The requirements for a circuit to oscillate are that the loop gain of the circuit must be ≥ 1 and the loop phase shift must be zero. This implies positive feedback. For sinusoidal output from an oscillator these requirements should be satisfied at one frequency only.



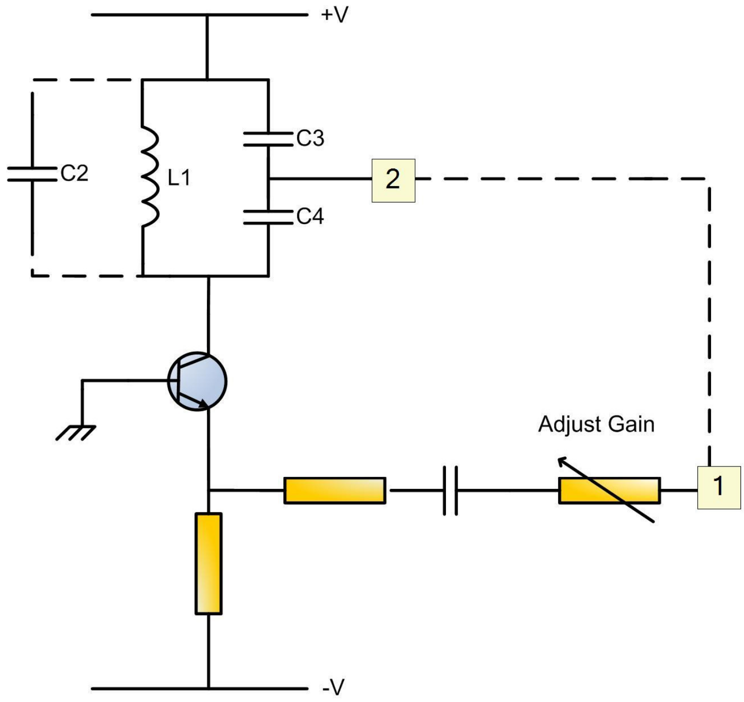
Under these circumstances, the input can be removed and there will be an output signal still present - the circuit oscillates.

In this Practical you will firstly set up the tuned amplifier, driven by the sweep source, to give a gain of just less than unity. This is the open loop gain. You will then remove the source, close the loop and see if the circuit oscillates.

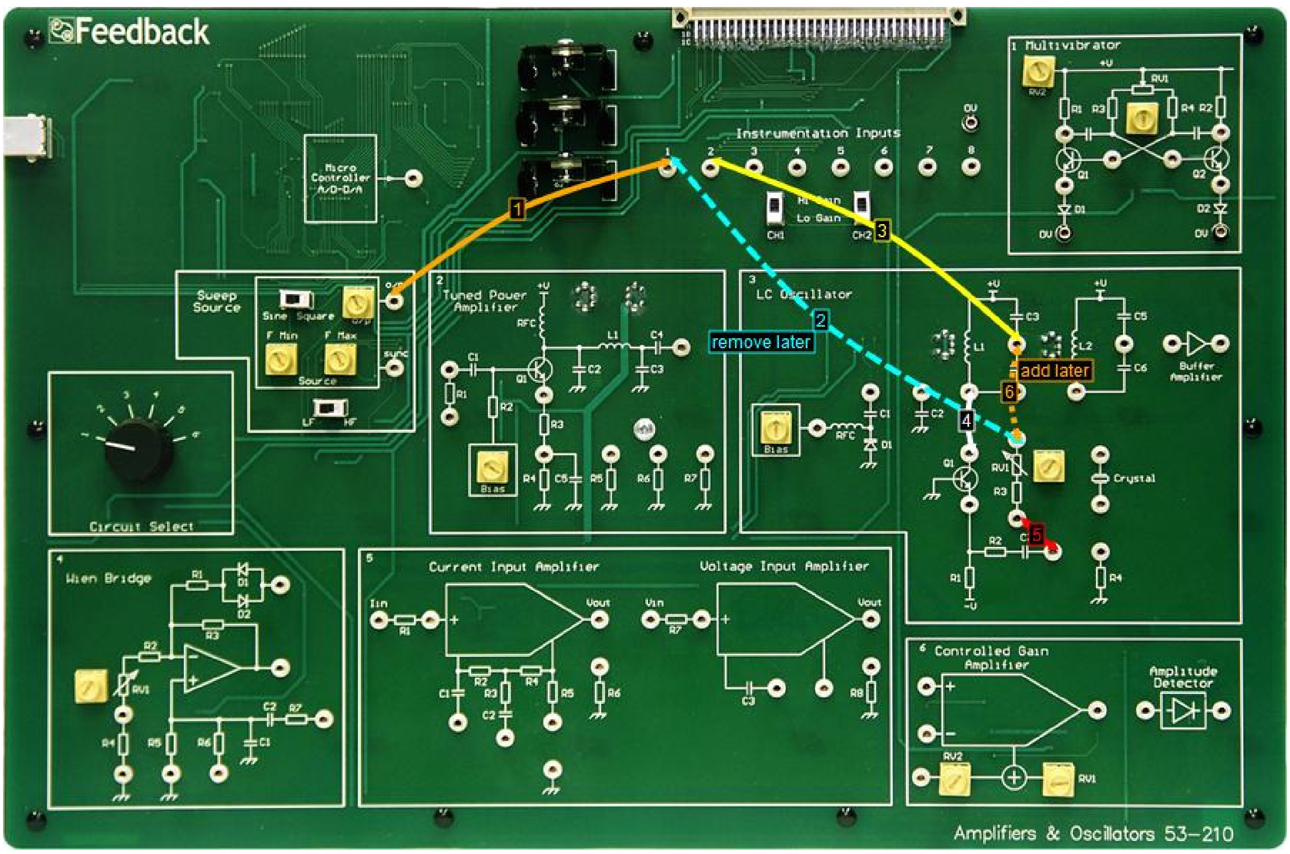
You will then increase the gain and examine the circuit again for oscillation. If achieved, you will measure the frequency of oscillation and the amplitude of the output.

Finally, you will break the loop and reconnect the external sweep source. Then you will measure the gain and phase required for the circuit just to oscillate.

Block Diagram



Make Connections Diagram



Practical 3: Requirements for Oscillation

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Open the Gain Phase Analyser (GPA). The **Set Min Freq** button on the GPA is already selected by default.

Use the **FMin** control of the **Sweep Source** to set the minimum sweep frequency to approximately 1.5MHz.

Click on the **Set Max Freq** button on the GPA and use the **FMax** control on the Sweep Source to approximately 2.2MHz.

Now, click the **Plot** button on the GPA.

Use the **RV1** control to adjust the peak amplitude of the response to approximately –1dB (i.e. slightly less than zero).

Use the GPA cursor to determine the frequency at which the phase goes through zero. Note this frequency.

This has now adjusted the amplifier to have a gain of just less than unity at the frequency of resonance.

Close the GPA.

Now remove connection 2 and insert connection 6, as shown in the Make Connections diagram. This now makes the connection between output and input (shown dotted on the Perform Practical diagram) and thus closes the loop.

Open the oscilloscope and confirm that there is no signal present at the output (i.e. the circuit is NOT oscillating), as the loop gain is less than one.

Now increase the gain of the amplifier by slowly decreasing RV1 (turning it counter-clockwise) until the circuit suddenly gives an output. It is now producing an output signal for no signal input; i.e. it is oscillating.

Use RV1 to adjust the output voltage of the oscillator to approximately 0.4V pk-pk. Open the frequency counter and measure the frequency of oscillation.

Compare this frequency with the resonant frequency that you measured for the amplifier.

Close the oscilloscope and the frequency counter.

Now break the feedback loop by removing connection 6 and restoring connection 2 in order to re-measure the open loop amplifier gain.

Open the GPA and use the cursor to measure the amplitude and frequency at the point of resonance. You should notice that the gain is now just slightly greater than unity (>0dB) at the frequency of resonance - the requirement for oscillation at this frequency. At all other frequencies the gain is less than unity and thus the requirements for oscillation are only met at one frequency, giving a sinusoidal (single frequency) output.

Practical 4: Stability

Objectives and Background

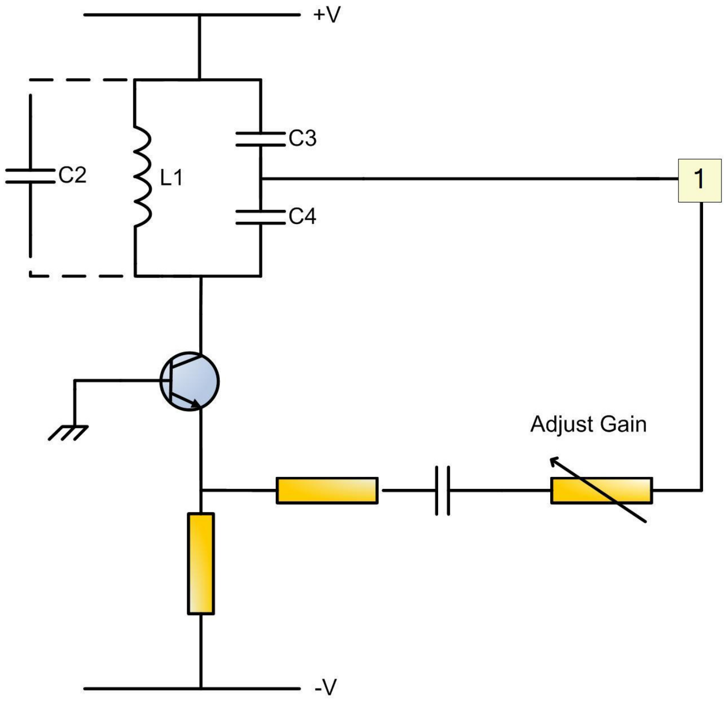
As you have found from Practical 3, the amplitude of the output from an oscillator is dependant on the loop gain of the circuit.

With the loop gain less than one the circuit will not oscillate
With the loop gain exactly one the circuit just oscillates, but with very low amplitude
With the loop gain greater than one the circuit oscillates and the amplitude quickly builds up until the circuit becomes non-linear.

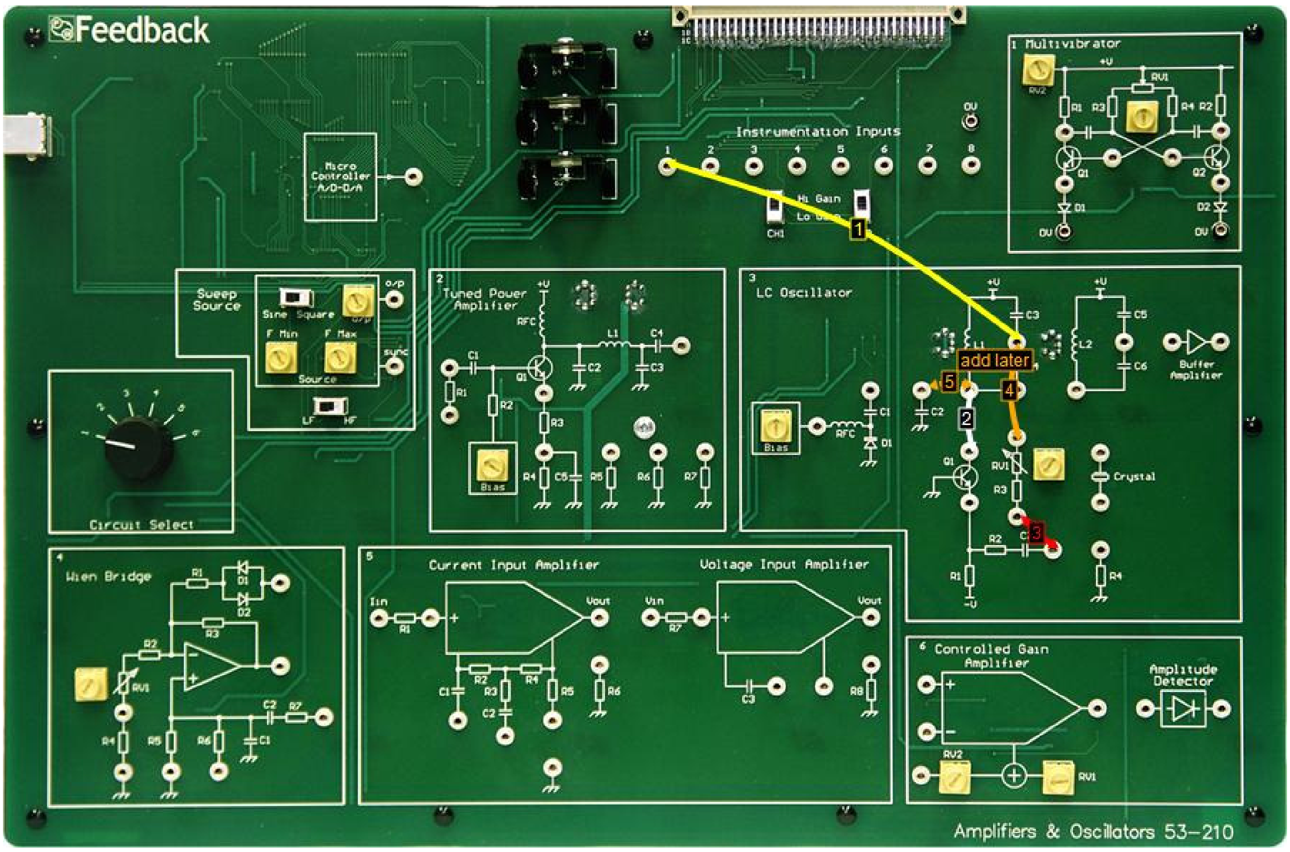
With the loop gain set for one, or just slightly over, any change in loop gain due to device changes (with temperature or voltage, for example) is likely to cause a change in output signal amplitude. Most practical oscillators have their loop gain set above one and they rely on the onset of non-linearities of the gain transfer characteristic to stabilise the amplitude of the output.

In this Practical you will investigate the effect that loop gain has on the output signal amplitude and its stability and you will see the effect of setting the gain greater than one and using the non-linearities of the circuit to stabilise the amplitude. You will also see that the frequency of oscillation is output amplitude dependant.

Block Diagram



Make Connections Diagram



Practical 4: Stability

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **CH1** gain switch, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set the CH1 switch to **Lo Gain**. The oscillator circuit can produce several volts peak-to-peak output, which is too high for the RAT input. Setting the CH1 switch to Lo Gain introduces x10 attenuation before the RAT.

Set the **RV1** control in the **LC Oscillator** circuit block to half scale.

Open the oscilloscope and adjust the RV1 control until the circuit just starts to oscillate.

Open the spectrum analyser and note that the spectrum is essentially just a single frequency, signifying that the output is a sine wave.

Open the voltmeter and note the amplitude of the output signal.

Identify the connecting lead between the collector of the transistor and the tuned circuit (connection 2 on the Make Connections diagram). Lightly hold this lead and notice what happens to the output signal. Note that the oscillation is very sensitive to outside influence and it is easy to stop the circuit from functioning.

Now, adjust RV1 to give more loop gain and produce a higher output amplitude. Note what happens to the spectrum analyser display.

Touch the collector connecting lead again and note that the circuit is much less likely to stop oscillation or change in amplitude. However the spectrum shows that the output is less sinusoidal. It is the onset of this distortion that stabilises the loop gain and makes the operation more reliable - but at the expense of spectral purity.

Adjust RV1 for just oscillation again. Open the frequency counter and note the frequency of oscillation and the amplitude of the output signal.

Now adjust RV1 to give maximum output amplitude, but with distortion. Note the frequency of oscillation now. Not only has the increase in amplitude caused distortion of the output but you should notice that the frequency of oscillation has changed. This is due to changes in transistor characteristics (internal capacitance) that are dependant on its operating conditions. In general, it is common practice to design oscillators to operate with as low a loop gain (and thus output amplitude and stable frequency) as is possible, consistent with reliable operation, and to use an additional amplification to give the required final signal amplitude.

The LC Oscillator (2)

Objectives

To see the effect on oscillation and frequency that loading has on an oscillator circuit

To investigate the effect of connecting a buffer stage between oscillator and load

To appreciate the operation of a variable capacitance diode

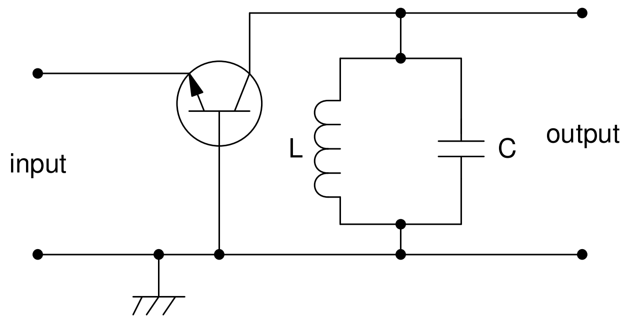
To investigate the operation of an LC oscillator circuit tuned by a varicap diode

Tuned Amplifiers

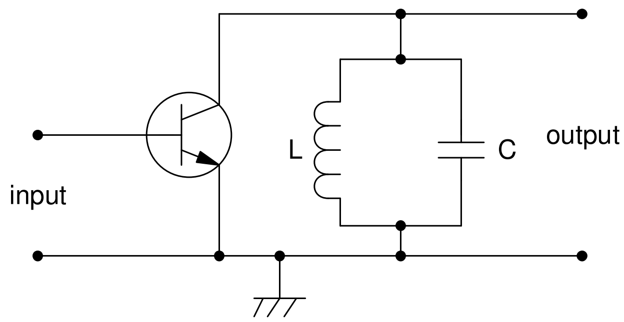
A tuned amplifier is one that has an LC tuned circuit, generally, as the load for the amplifying device.

For a bipolar transistor amplifier, the most common connections for the device used in a tuned amplifier are common emitter or common base. In each of these connections the output terminal of the transistor is the collector. The tuned circuit is thus most commonly connected in the collector circuit.

Simplified circuits, excluding any power supplies or components required to bias the transistor correctly, are shown below.



Common Base Connection



Common Emitter Connection

In both cases, above, the impedance of the parallel LC tuned circuit (ignoring any component losses and resistances) is given by:

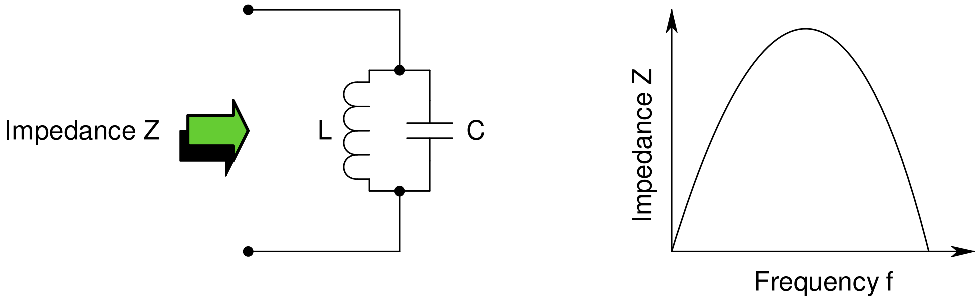
$$Z = \frac{j\omega L}{1 - \omega^2 LC}$$

This is frequency (ω) dependant. It will be a maximum when

$$\omega^2 LC = 1$$

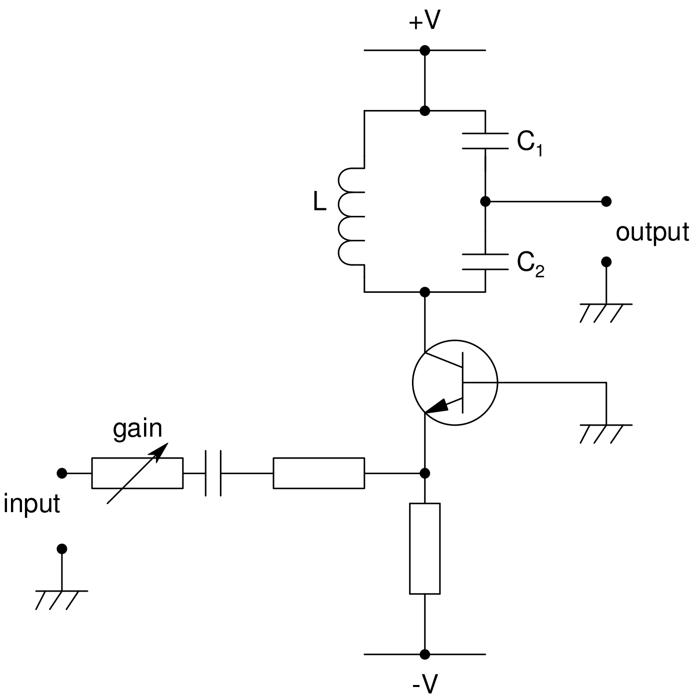
Thus the gain of the amplifiers will be a maximum at this frequency. This frequency is the **resonant frequency** of the tuned circuit.

At any frequency away from the resonant frequency the impedance of the tuned circuit will drop, as shown below.



Thus the frequency response of the amplifier will be of a similar form:

A typical common base circuit (that is used on the workboard), complete with bias resistors, capacitors, etc, is shown below.



The two capacitors in the tuned circuit form, effectively, a capacitive potential divider. This is required for the purposes of the assignment. The equivalent value (C) of these two capacitors is given by their series combination.

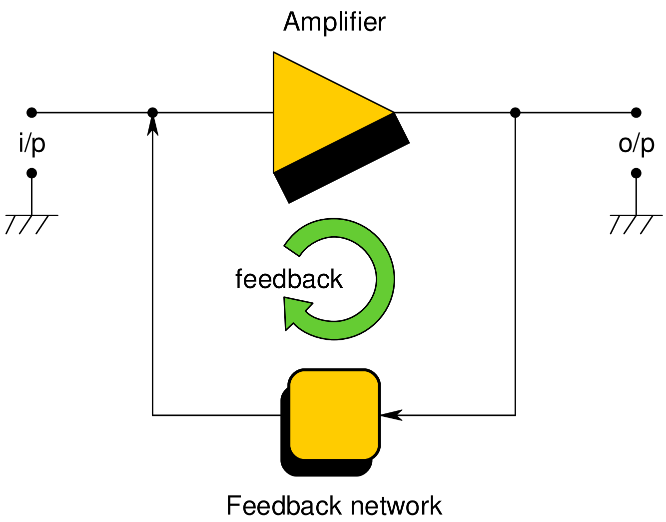
$$C = \frac{C_1 C_2}{C_1 + C_2}$$

Requirements for Oscillation

An oscillator is a circuit that produces an output signal for no input signal.

An alternative way of thinking of an oscillator is that it is a circuit that converts dc power (which comes from the dc supply to the circuit) into signal power that is available at the output terminals.

Consider an amplifier with feedback applied from the output back to the input, as shown in the following diagram.



The signal fed back can be such that it either helps to cancel or reinforce the original input signal, depending on the **phase** of the feedback signal.

Signals that are fed back in phase with the original input signal will reinforce its effect

Signals that are fed back in anti-phase with the original input signal will reduce its effect

This gives rise to two forms of feedback:

Signals that are fed back in phase with the original input signal give **positive feedback**

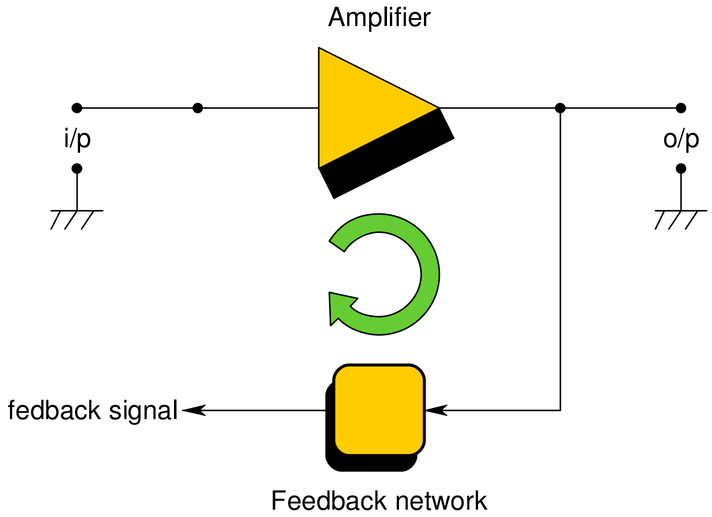
Signals that are fed back in anti-phase with the original input signal give **negative feedback**

An oscillator requires positive feedback to be applied.

The combination of the amplifier and the feedback network is known as the **feedback loop**. For positive feedback to be applied the signal fed back round the loop must be such that it is **in phase** with the original input signal. This means that the **total** phase shift, first through the amplifier and then through the feedback network, must be 0 degrees. This is usually referred to as **the loop phase shift**.

Having determined that loop phase shift must be zero to get full signal reinforcement, the next question is “how much signal needs to be fed back to get sustained oscillation?”

To answer this question consider the circuit above, but with the feedback loop ‘broken’, as shown in the diagram below.



Suppose the amplifier had a gain of 3 and the feedback network attenuated the signal by a factor of 5 (i.e. it has a ‘gain’ of 0.2). This means that the feedback signal will be 0.2 of the amplified signal, i.e. 0.2 x 3 times the original input signal: = 0.6 of the input amplitude.

The combination of the amplifier gain and the feedback network attenuation is called the **loop gain**.

To be able to maintain an output, the attenuation of the feedback network must at least be made up by the gain of the amplifier. That **means that the loop gain must be at least unity**.

In this instance, the feedback signal is less than the original input signal so, if the feedback loop were reconnected and the input signal removed, there would not be enough feedback to maintain an output. The loop gain is less than unity. It is usually much easier to change the gain of an amplifier than the attenuation of the feedback network thus, to get unity loop gain, in this instance the amplifier gain must be increased to at least 5.

Thus, the criteria for oscillation are:

the loop phase shift must be zero and

the loop gain must be at least one

The Varicap Diode

Varicap diodes are semiconductor devices that are widely used in the electronics industry and are used in many applications where a voltage controlled variable capacitance is required. Accordingly they are used in circuits including voltage controlled oscillators, and filters.

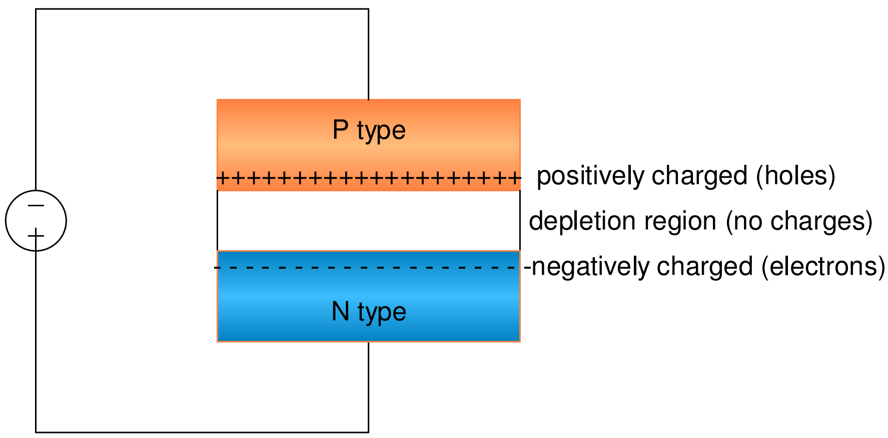
Their operation is based on the fact that a reverse biased PN junction acts as a small variable capacitor. Varying the reverse voltage changes the capacitance. Although any ordinary diode may be used in this way, diodes manufactured specifically for this purpose can offer controlled and higher levels of capacitance.

Concept

The varicap diode employs a standard PN junction. This consists of a region of P type material and a region of N type material. The N type region has a surfeit of electrons, whereas the P type region has a shortage; what are termed holes are generated. These are empty spaces available for an electron within the crystal lattice of the semiconductor. Both electrons and holes can move around the lattice and, in this way, current flows if there is a general drift in one particular direction.

In the region where the P and N type semiconductor adjoin it is found that the electrons and holes combine and there are no carriers available to give rise to any current flow. It is this gap that is used as the dielectric between the two plates of the capacitor; the two plates being formed by the boundary of where the carriers are available to conduct electricity. As the capacitance of a capacitor is related to the distance between the two plates, it is possible to change the capacitance by varying the width of the depletion region.

Reversed biased PN junction



If a voltage is applied across the device, the width of the depletion layer changes. The greater the level of reverse bias that is placed across the diode, the greater the depletion region becomes and the further apart the "plates" become, and the smaller the capacitance across the diode. If a forward bias is placed across the diode, the depletion region reduces and eventually conduction takes place.

Varicap diodes are always operated under reverse bias conditions, and in this way there is no conduction. They are effectively voltage controlled capacitors.

Parameters

The actual capacitance range that is obtained depends upon a number of factors. One is the area of the junction. Another is the width of the depletion region for a given voltage. This is governed by the doping concentration and it is normally adjusted to give a relatively abrupt junction which results in a greater capacitance change.

Diodes typically operate with reverse bias ranging from around a couple of volts up to 20 volts and higher. Some may even operate up to as much as 60 volts, although at the top end of the range comparatively little change in capacitance is seen.

Characteristics

The most important characteristics of the diode are its capacitance and the range of capacitance that can be achieved. Normally two voltage points are specified, one at the top of the range and the other near the bottom at the minimum useable voltage. It is obviously important to select a diode that combines the correct capacitance range for the available tuning voltage range. The higher voltage specified is normally the maximum reverse bias and this should not be exceeded otherwise breakdown may occur.

An important characteristic of any varicap diode is its Q. This is particularly important in a number of applications. For oscillators used in frequency synthesizers it affects the noise performance. High Q diodes enable a higher Q tuned circuit to be achieved, and in turn this reduces the phase noise produced by the circuit. For filters the Q is again very important. A high Q diode will enable the filter to give a sharper response, whereas a low Q diode will increase the losses.

Some varicap diodes may be referred to as abrupt and hyper-abrupt types. The term refers to the junction where the change between P and N types is either abrupt or very/hyper abrupt. With a very sharp junction, these diodes offer a relatively large percentage change in capacitance. They are particularly useful when oscillators or filters need to be swept over a large frequency range.

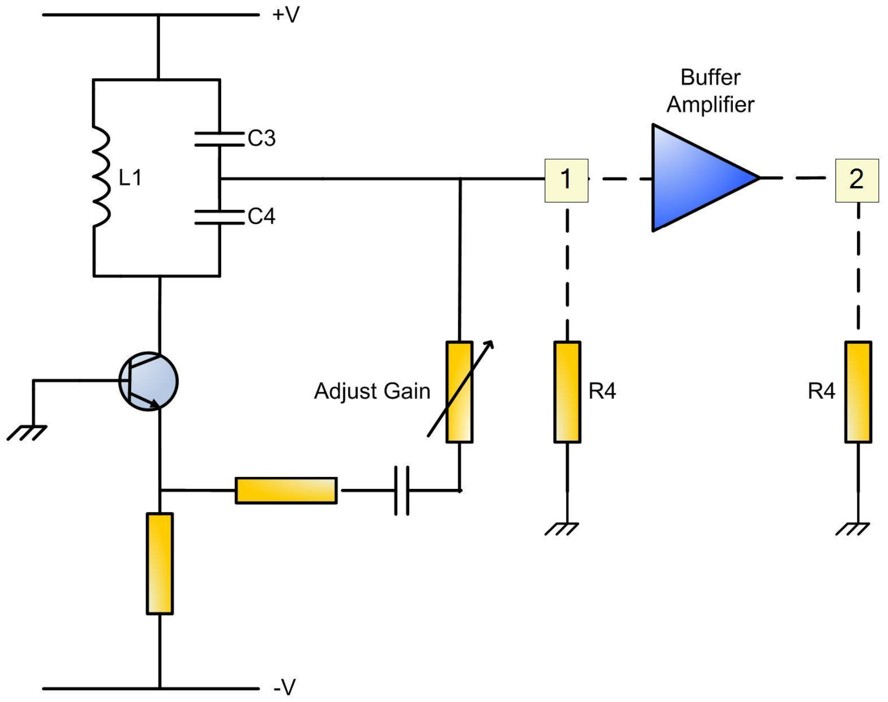
Practical 1: Loading and Buffering

Objectives and Background

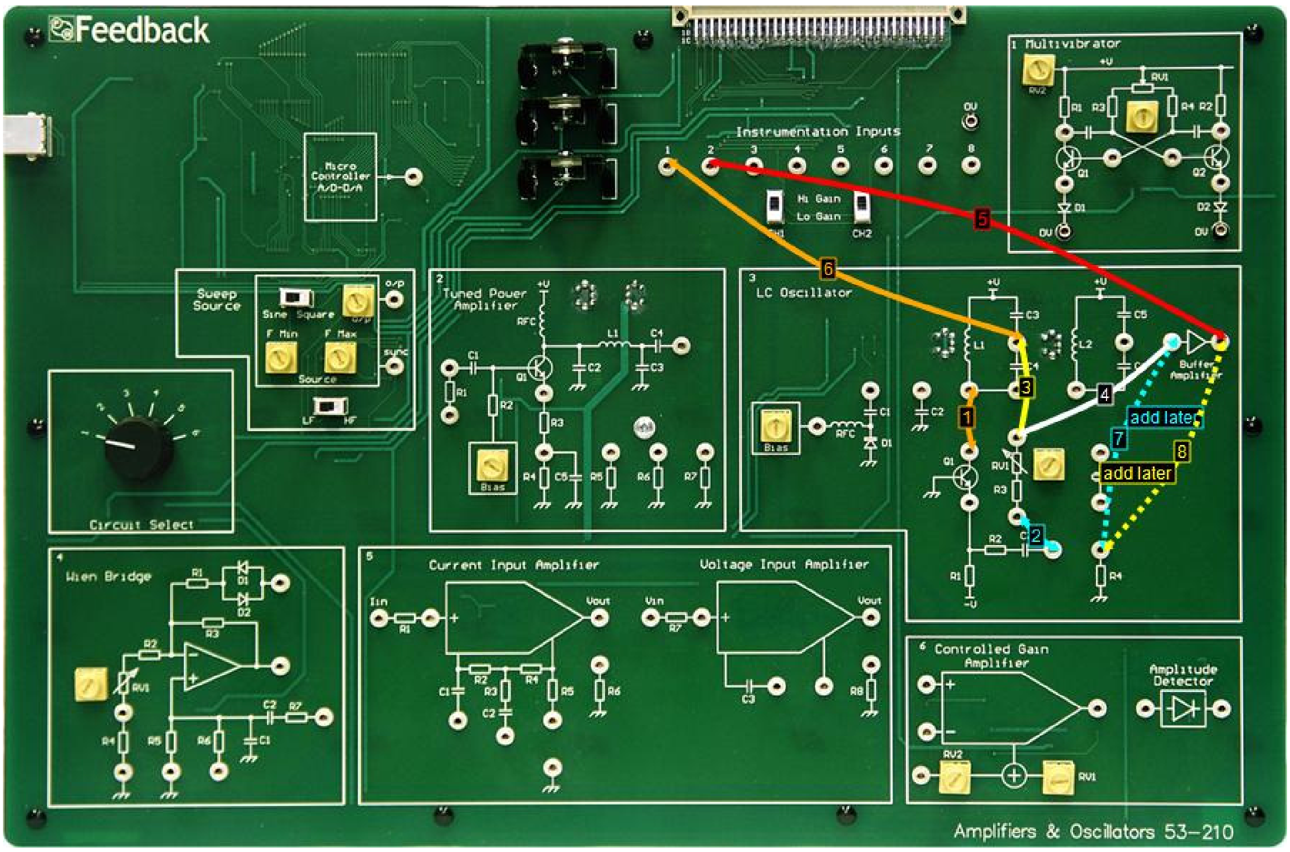
In this Practical you will investigate the effects of loading the oscillator circuit with a resistor. This simulates the effect of connecting other stages to the output of the circuit. You will see that the loop gain of the oscillator can be affected, as can its frequency of oscillation.

You will then connect a buffer amplifier to the output of the oscillator and investigate the effect of loading the combined circuit.

Block Diagram



Make Connections Diagram



Practical 1: Loading and Buffering

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **LC Oscillator** circuit block, located to the centre-right of the workboard.

Ensure that the **Circuit Select** switch, on the left of the workboard, is set to position **3**.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 switches to **Hi Gain**.

Set **RV1** on the LC Oscillator circuit to half scale.

Open the oscilloscope and the frequency counter. Adjust RV1 slowly counter-clockwise until the circuit starts to oscillate. You will now need to adjust the oscilloscope timebase to get a satisfactory display.

Set RV1 to give approximately 0.6V pk-pk oscillation at the output (the yellow probe on monitor point 1).

Note the amplitude of the signal at the output of the buffer amplifier. Use the **Overlay** function on the oscilloscope to compare the two amplitudes. You should see that the buffer amplifier has unity gain.

Note the frequency of oscillation.

Now connect **R4** to the output of the oscillator (monitor point 1) as shown by connection 7 on the Make Connections diagram.

Note what happens to the oscillator output signal. Adjust RV1 to bring the oscillator output back to approximately 0.6V pk-pk. Note the frequency of oscillation.

Remove R4 from circuit again and observe the output waveform and the frequency of oscillation. Reset RV1 to give 0.6V pk-pk oscillation again.

What you have just done is load the oscillator circuit with a fairly low value of resistance (R4 is 470Ω). You have seen that, if the oscillator is set up to give a sine wave output with no load, loading it can stop the circuit from oscillating. If the circuit is set up to give a sine wave output with the load connected then if the load is removed the output amplitude rises greatly and the output becomes very distorted. Now you will see what effect a buffer amplifier makes.

Transfer the load (R4) to the output of the buffer amplifier (monitor point 2) as shown by connection 8 in the Make Connections diagram. Note what happens to the oscillator output and also the frequency of oscillation.

In practice, it is always good design practice to follow an oscillator with a buffer stage to minimise any effects of load change.

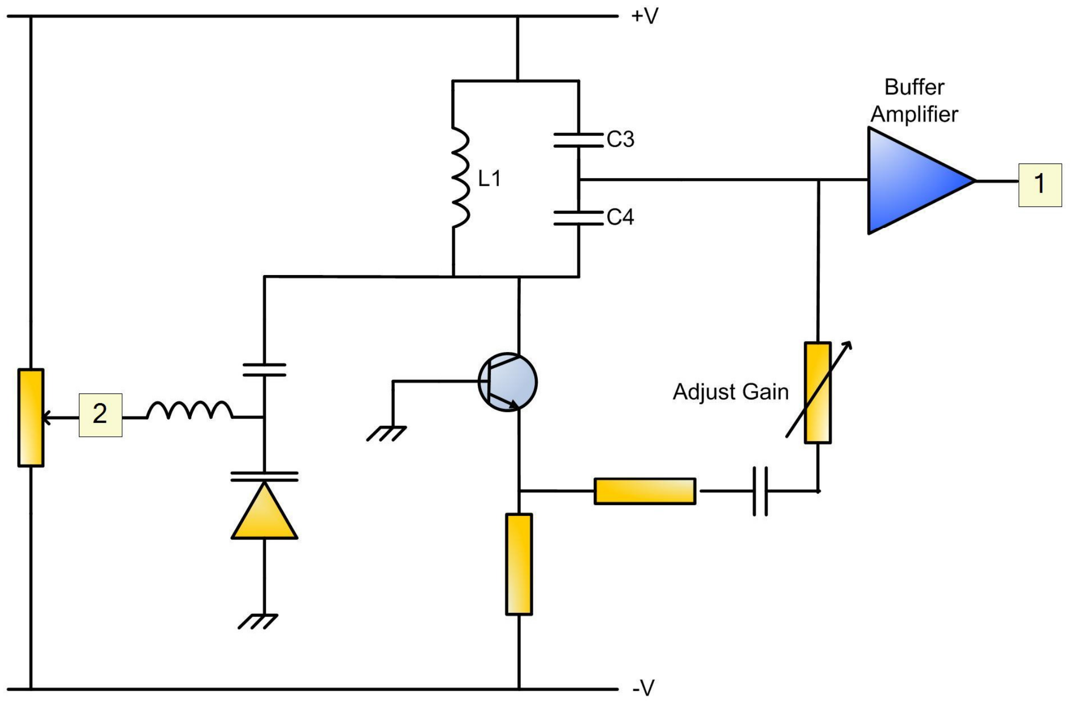
Practical 2: Varicap Diode Tuning

Objectives and Background

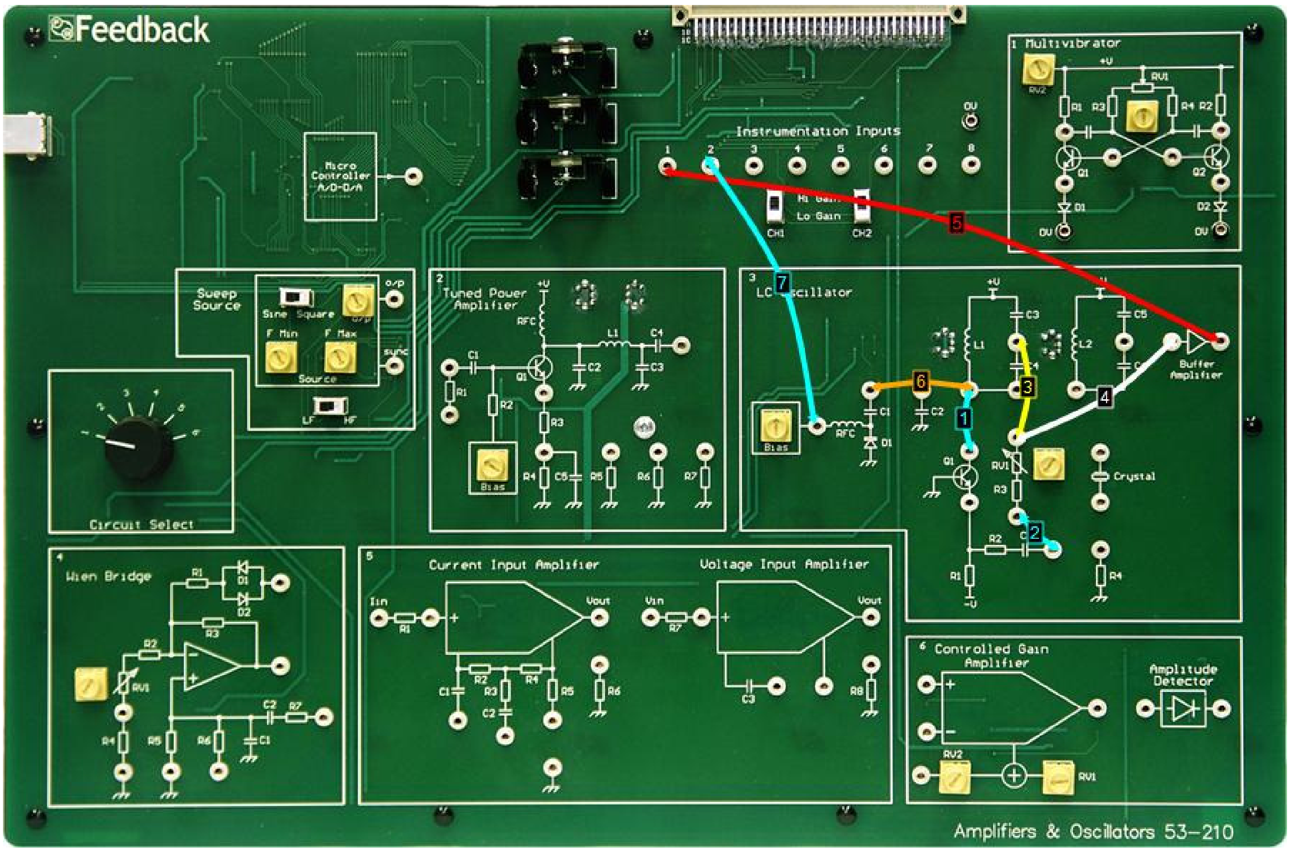
In this Practical you will investigate tuning the oscillator by means of a variable capacitance (varicap) diode fed by a variable voltage source.

You will plot the variation of frequency against tuning voltage.

Block Diagram



Make Connections Diagram



Practical 2: Varicap Diode Tuning

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that the **Circuit Select** switch, on the left of the workboard, is set to position **3**.

Set the **CH1** switch to **Hi Gain** and the **CH2** switch to **Lo Gain**.

Set **RV1** within the **LC Oscillator** circuit block to half scale.

Identify the variable capacitance diode (varicap) circuit on the left-hand side of the LC Oscillator circuit block. Set the **Bias** control to half scale.

Open the oscilloscope, the voltmeter and the frequency counter. Adjust RV1 slowly counter-clockwise until the circuit starts to oscillate. You will now need to adjust the oscilloscope timebase to get a satisfactory display.

Set RV1 to give approximately 0.6V pk-pk oscillation at the output (blue probe on monitor point 1).

Note the frequency of oscillation and the dc bias voltage on the varicap diode (yellow probe on monitor point 2), as displayed on the voltmeter.

Now vary the position of the **Bias** control and note the variation of frequency of oscillation with applied dc voltage. Open the spreadsheet and enter your results in the table.

Notice that the variation in frequency with voltage is not linear. This is typical for varicap frequency control.

The Crystal Oscillator

Objectives

To become familiar with the operation of a piezoelectric quartz crystal

To appreciate how a quartz crystal can be used to determine the frequency of operation of an oscillator

To investigate the use of a crystal at its fundamental frequency in an oscillator circuit

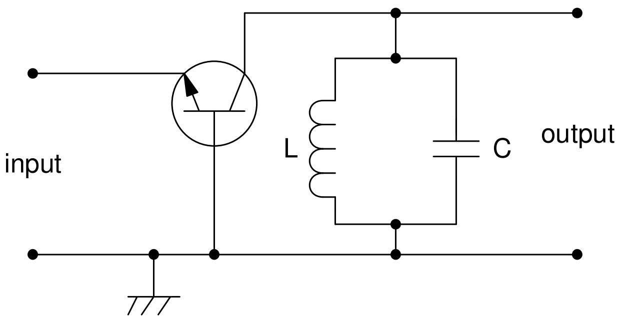
To investigate the operation as an oscillator of an crystal at its third overtone

Tuned Amplifiers

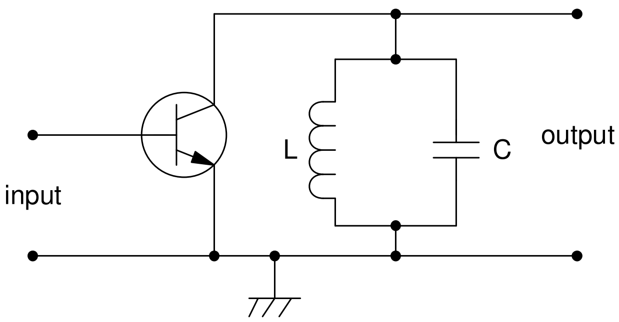
A tuned amplifier is one that has an LC tuned circuit, generally, as the load for the amplifying device.

For a bipolar transistor amplifier, the most common connections for the device used in a tuned amplifier are common emitter or common base. In each of these connections the output terminal of the transistor is the collector. The tuned circuit is thus most commonly connected in the collector circuit.

Simplified circuits, excluding any power supplies or components required to bias the transistor correctly, are shown below.



Common Base Connection



Common Emitter Connection

In both cases, above, the impedance of the parallel LC tuned circuit (ignoring any component losses and resistances) is given by:

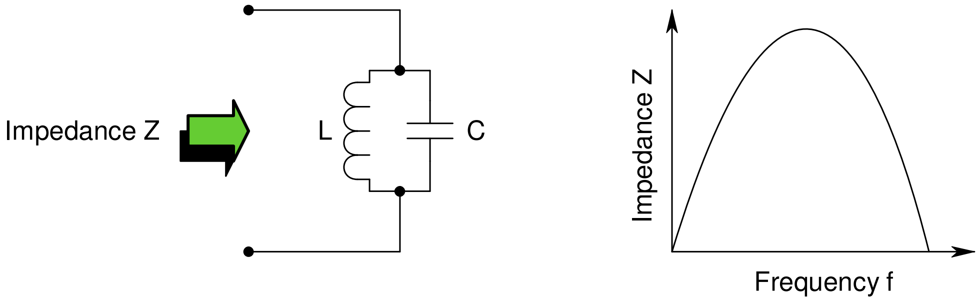
$$Z = \frac{j\omega L}{1-\omega^2 LC}$$

This is frequency (ω) dependant. It will be a maximum when

$$\omega^2 LC = 1$$

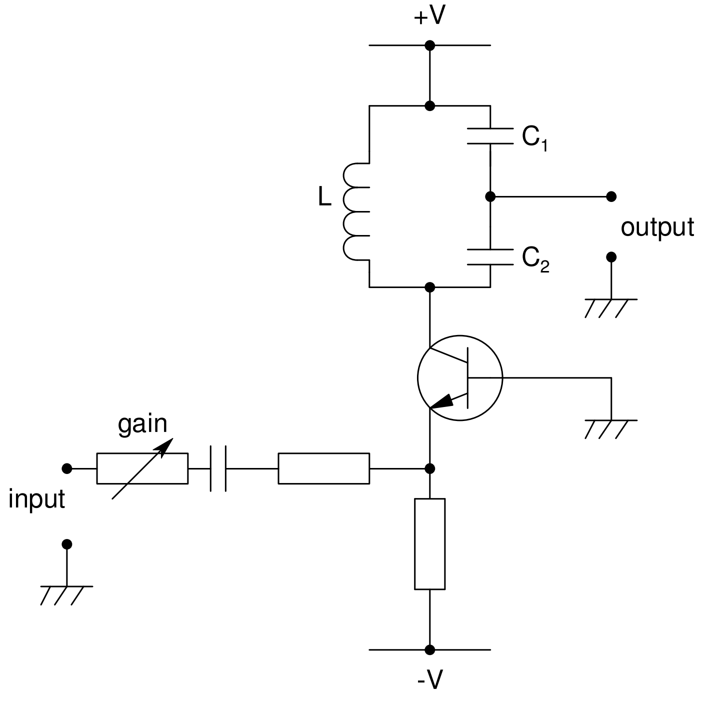
Thus the gain of the amplifiers will be a maximum at this frequency. This frequency is the **resonant frequency** of the tuned circuit.

At any frequency away from the resonant frequency the impedance of the tuned circuit will drop, as shown below.



Thus the frequency response of the amplifier will be of a similar form:

A typical common base circuit (that is used on the workboard), complete with bias resistors, capacitors, etc, is shown below.



The two capacitors in the tuned circuit form, effectively, a capacitive potential divider. This is required for the purposes of the assignment. The equivalent value (C) of these two capacitors is given by their series combination.

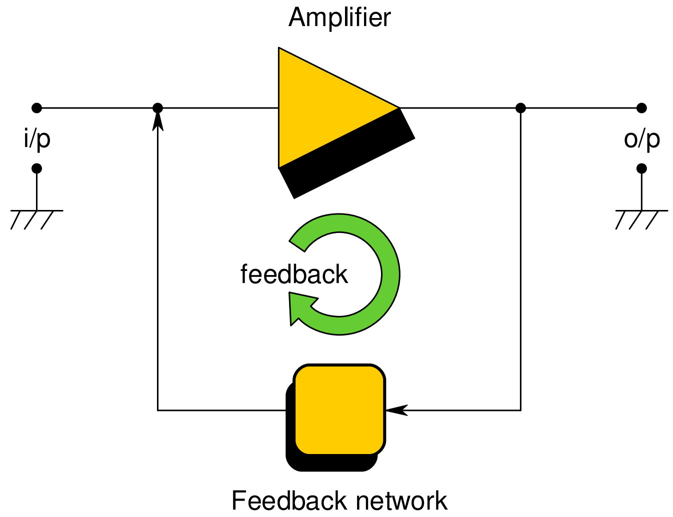
$$C = \frac{C_1 C_2}{C_1 + C_2}$$

Requirements for Oscillation

An oscillator is a circuit that produces an output signal for no input signal.

An alternative way of thinking of an oscillator is that it is a circuit that converts dc power (which comes from the dc supply to the circuit) into signal power that is available at the output terminals.

Consider an amplifier with feedback applied from the output back to the input, as shown in the following diagram.



The signal fed back can be such that it either helps to cancel or reinforce the original input signal, depending on the **phase** of the feedback signal.

Signals that are fed back in phase with the original input signal will reinforce its effect

Signals that are fed back in anti-phase with the original input signal will reduce its effect

This gives rise to two forms of feedback:

Signals that are fed back in phase with the original input signal give **positive feedback**

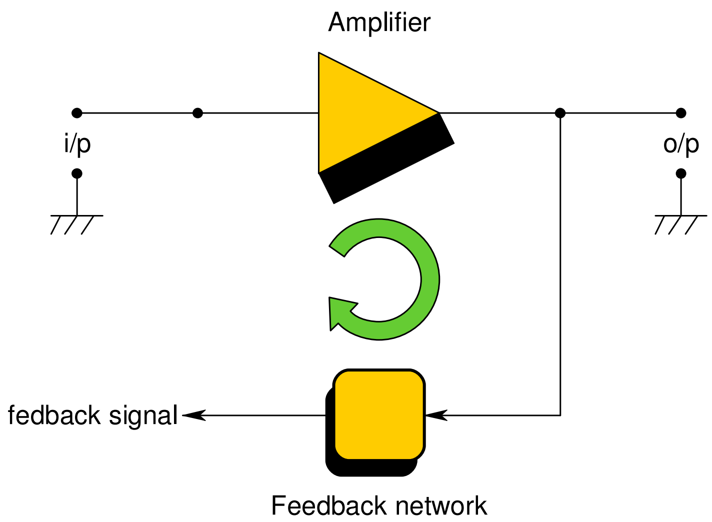
Signals that are fed back in anti-phase with the original input signal give **negative feedback**

An oscillator requires positive feedback to be applied.

The combination of the amplifier and the feedback network is known as the **feedback loop**. For positive feedback to be applied the signal fed back round the loop must be such that it is **in phase** with the original input signal. This means that the **total** phase shift, first through the amplifier and then through the feedback network, must be 0 degrees. This is usually referred to as **the loop phase shift**.

Having determined that loop phase shift must be zero to get full signal reinforcement, the next question is “how much signal needs to be fed back to get sustained oscillation?”

To answer this question consider the circuit above, but with the feedback loop ‘broken’, as shown in the diagram below.



Suppose the amplifier had a gain of 3 and the feedback network attenuated the signal by a factor of 5 (i.e. it has a ‘gain’ of 0.2). This means that the feedback signal will be 0.2 of the amplified signal, i.e. 0.2 x 3 times the original input signal: = 0.6 of the input amplitude.

The combination of the amplifier gain and the feedback network attenuation is called the **loop gain**.

To be able to maintain an output, the attenuation of the feedback network must at least be made up by the gain of the amplifier. That **means that the loop gain must be at least unity**.

In this instance, the feedback signal is less than the original input signal so, if the feedback loop were reconnected and the input signal removed, there would not be enough feedback to maintain an output. The loop gain is less than unity. It is usually much easier to change the gain of an amplifier than the attenuation of the feedback network thus, to get unity loop gain, in this instance the amplifier gain must be increased to at least 5.

Thus, the criteria for oscillation are:

the loop phase shift must be zero and

the loop gain must be at least one

Crystal Oscillators

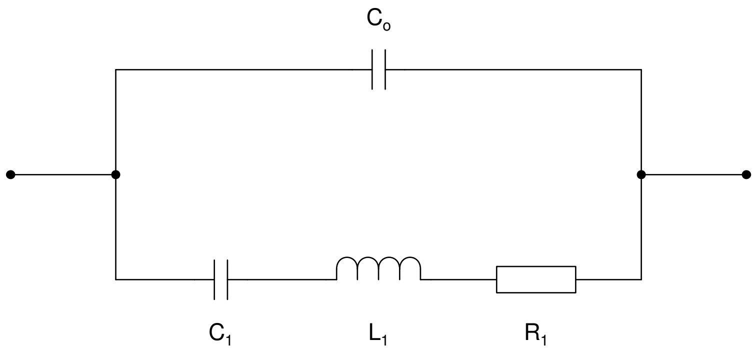
Certain crystals are called piezoelectric when they exhibit a relationship between mechanical strain (tension or compression) and voltage across their surfaces. Specifically, when compressed or pulled, a piezoelectric crystal will build up alternate charges on opposite faces, thus acting like a capacitor with an applied voltage. A current, called piezoelectricity, can then be generated between the faces. On the other hand, when subjected to an external voltage, the crystal will expand or contract accordingly.

This effect is put to use in several ways, the most common of which is in quartz crystal oscillators. When a crystal of quartz is correctly cut and mounted, it can be made to bend if a voltage is applied to it. When the voltage is removed the quartz crystal will generate a voltage as it returns to its original shape. The result is that the crystal behaves like a circuit composed of a capacitor, an inductor and a resistor. This is equivalent to a tuned circuit, with a resonant frequency determined by the dimensions and properties of the crystal. The crystal will have a very precise resonant frequency and thus very precise oscillators may be designed using crystals in circuit instead of (or as well as) a conventional LC circuit.

Such crystal oscillator circuits are common in communication electronics, where the generation of precise frequencies is important. Also, every computer has at least one clock frequency which is generated by a quartz crystal. Quartz has the advantage that its dimensions change very little with temperature, hence a crystal oscillator can be made very temperature stable.

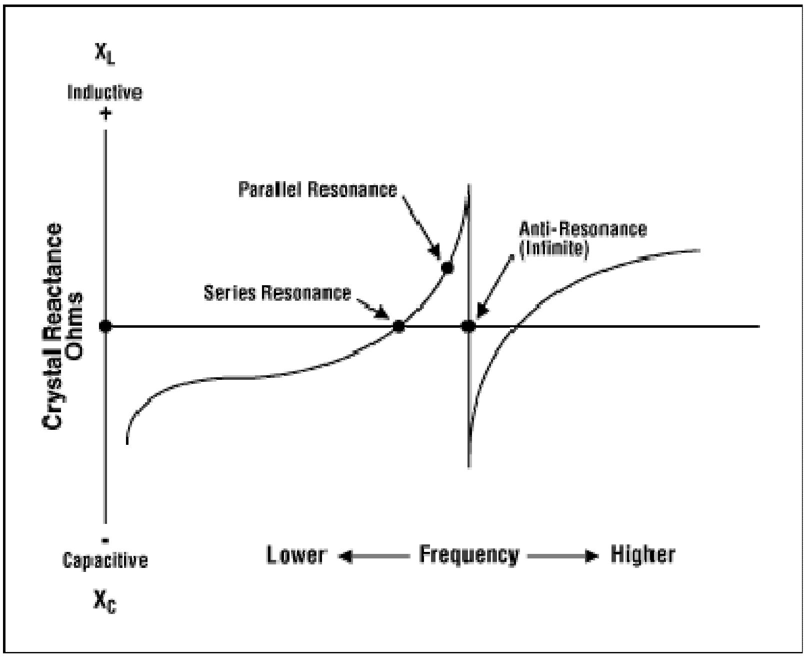
Quartz crystals for oscillators can be made for frequencies from a few tens of kHz to several tens of MHz.

The equivalent circuit for a quartz crystal is shown below.

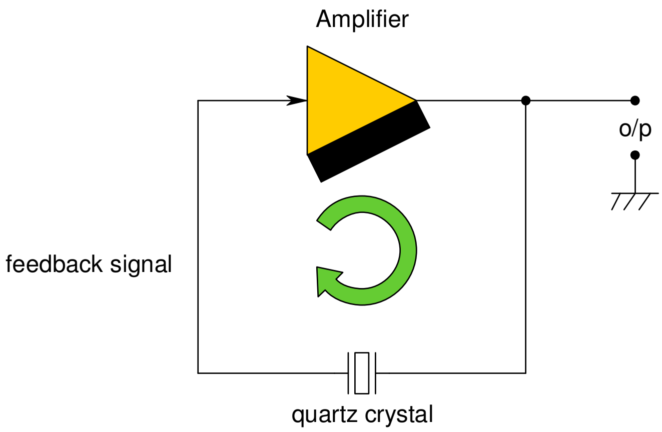


As you can see, there is a series resonant branch comprising C_1, L_1 and R_1. These are really the electrical equivalents of the mechanical properties of the crystal: C_1 represents the compliance of the crystal, L_1 its inertia and R_1 the losses due to lattice friction within the crystal. The electrical equivalent values are an extremely small value for C_1 (parts of a pF), a large value for L_1 (perhaps several Henries) and, as the losses are very low, a very low value for R_1 (parts of an Ohm). This means that the Q of the circuit is very high (much higher than a conventional LCR resonant circuit) and thus its resonant frequency is very precisely defined.

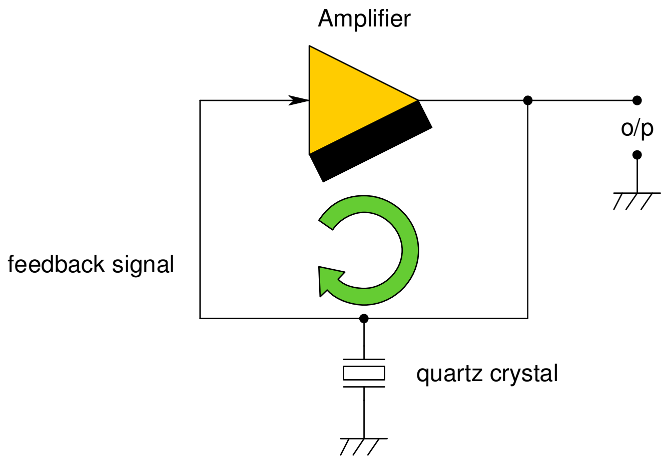
There is also the parallel branch of C_2. This is a real capacitor – the mounting capacitance of the electrodes and connection wires of the crystal – and is a few pF in practice. There will be a second parallel resonant frequency of the total crystal assembly, including this capacitance. These two resonances are very close together, with the series resonance being generally only a kHz or two below the parallel resonance. A typical graph of crystal reactance (impedance) with frequency is shown below:



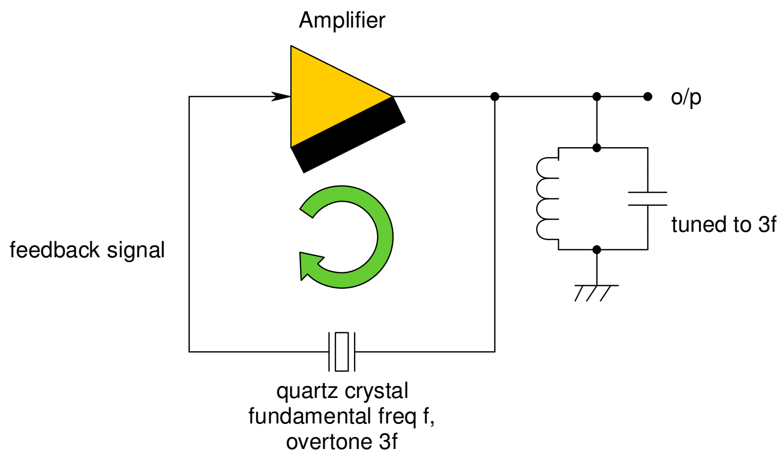
An oscillator circuit may use either one of the resonant forms. If the crystal is used as a series component in the feedback loop it will have very low impedance at its series resonant frequency, and thus only signals at this frequency will be fed back. An example is shown below:



At its parallel resonant frequency the crystal will have almost infinite impedance. If it is used in circuit as a load, only at its parallel resonant frequency will there be developed significant voltage across it. This is illustrated below:



The crystal can be made to resonate at (or close to) multiples of its fundamental frequency of oscillation. These are called its **overtone** frequencies, and are generally at odd multiples of the fundamental (3x, 5x, 7x, etc). For frequencies above a few tens of MHz, the crystal dimensions required (especially its thickness) become very small and the device becomes mechanically fragile and the crystal is very difficult to manufacture. Thus an overtone mode of operation is normally used for oscillators above a few tens of MHz. To ensure that the required overtone is selected, additional LC circuits, resonant at that overtone, are used. This is shown schematically below:

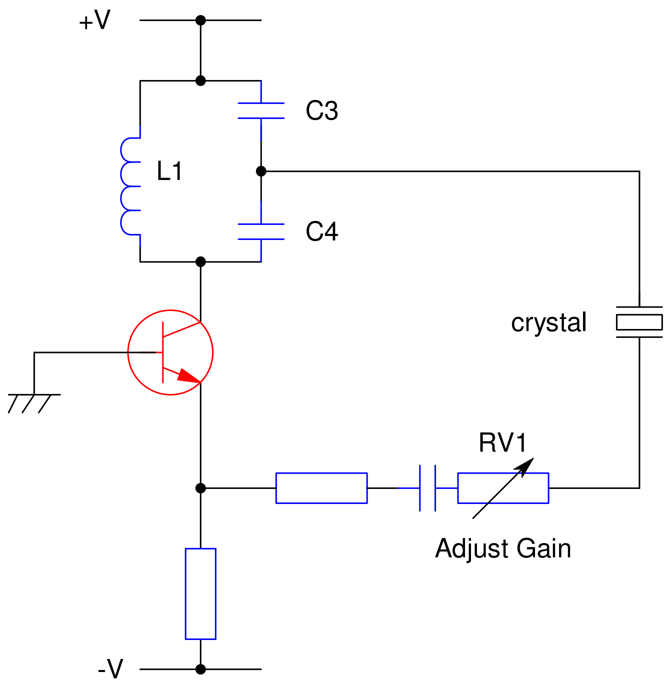


Practical 1: Fundamental Frequency Operation

Objectives and Background

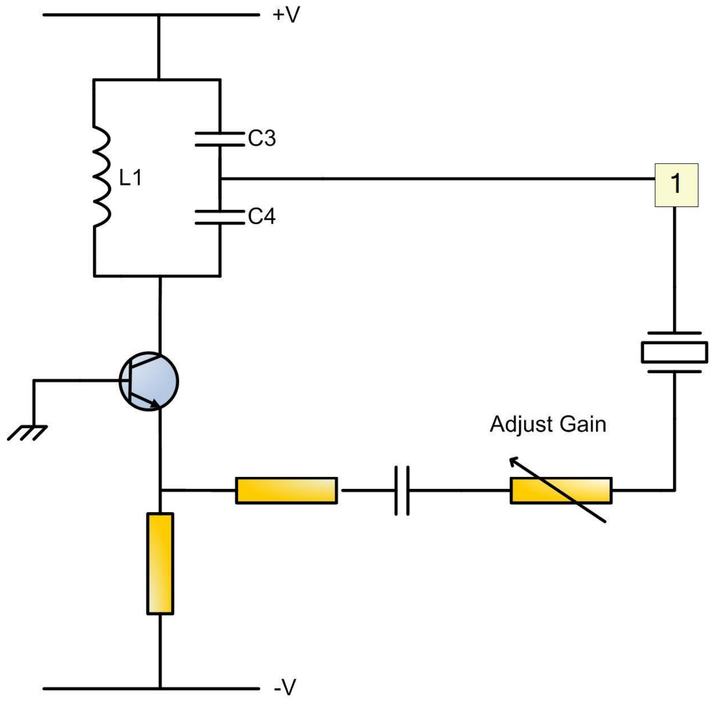
In this Practical you will investigate the operation of a crystal oscillator at its fundamental frequency of operation.

The circuit that you will be using is based on a tuned amplifier (operating at approximately 1.8MHz) with a quartz crystal in series in the feedback path. The crystal will have almost zero impedance at its fundamental series mode frequency of oscillation. The circuit is as shown below:

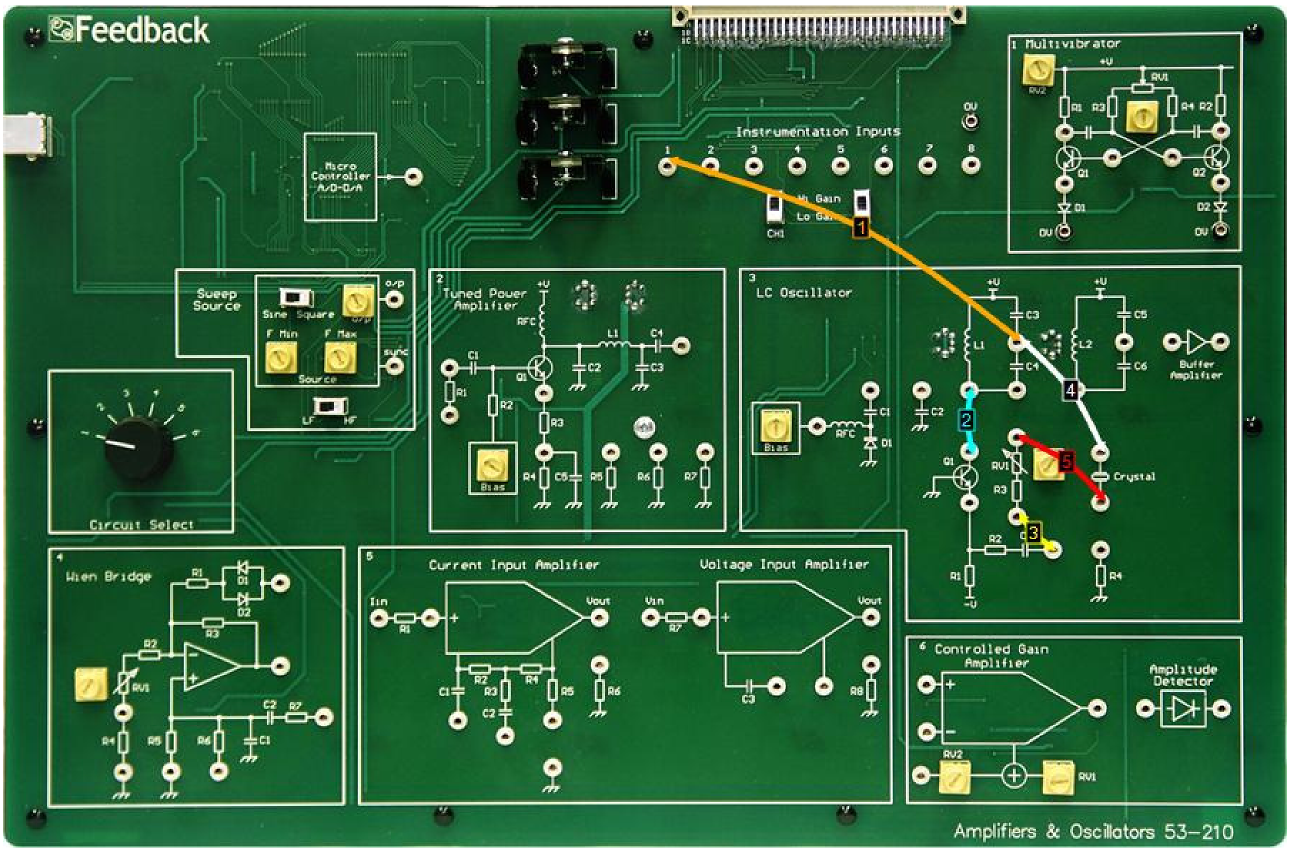


You will adjust the loop gain for sinusoidal oscillation and you will measure the frequency of oscillation.

Block Diagram



Make Connections Diagram



Practical 1: Fundamental Frequency Operation

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **LC Oscillator** circuit block, located to the centre-right of the workboard.

Ensure that the **Circuit Select** switch, on the left of the workboard, is set to position **3**.

Identify the **CH1** gain switch, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set the CH1 switch to **Lo Gain**. The oscillator circuit can produce several volts peak-to-peak output, which is too high for the RAT input. Setting the CH1 switch to Lo Gain introduces x10 attenuation before the RAT.

Set the **RV1** control within the LC Oscillator circuit block to half scale.

Open the oscilloscope and adjust the RV1 control until the circuit just starts to oscillate. Note this position; you will be comparing the position needed in Practical 2 with it.

Open the spectrum analyser and note that the spectrum is essentially just a single frequency, signifying that the output is a sine wave.

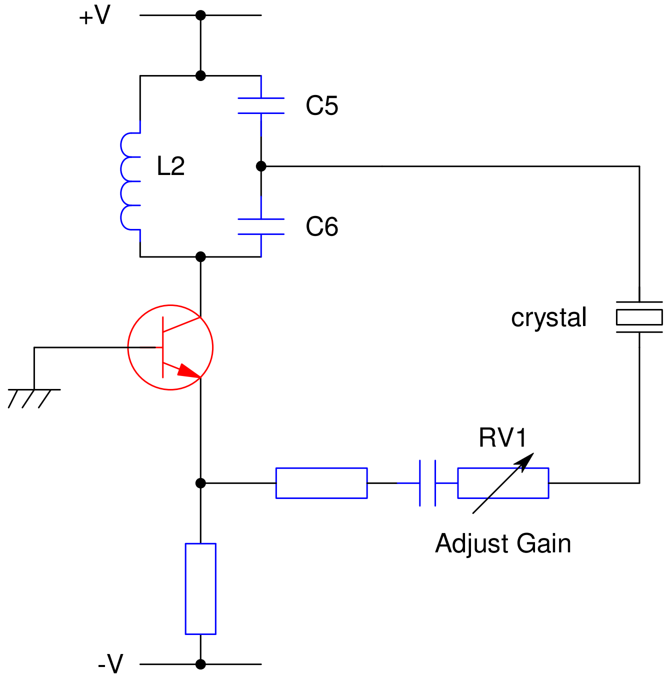
Open the counter and note the frequency of the output signal. This is the fundamental frequency of oscillation for the crystal.

Practical 2: Overtone Operation

Objectives and Background

In this Practical you will investigate the operation of a crystal oscillator at its third overtone frequency of operation.

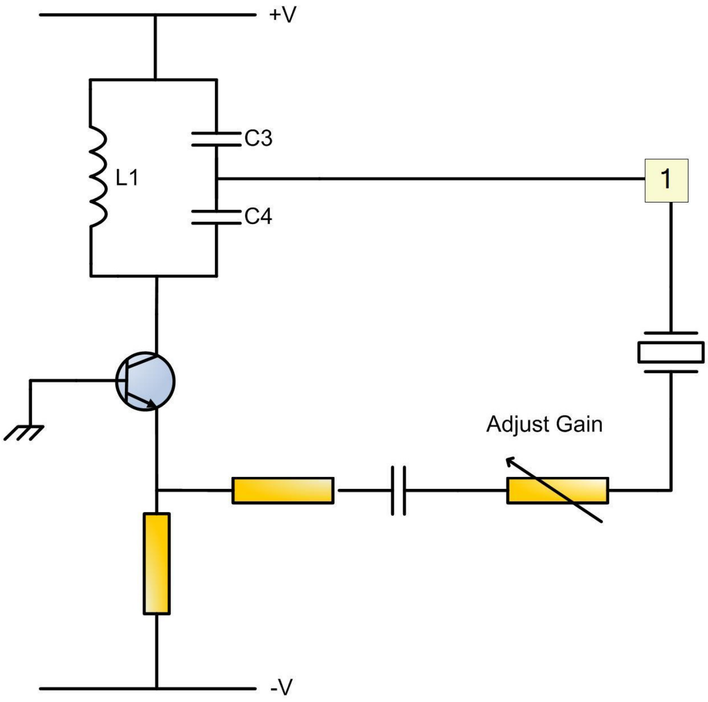
The circuit that you will be using is based on a tuned amplifier (operating at approximately three times the crystal fundamental frequency of oscillation) with a quartz crystal in series in the feedback path. The crystal will have a low impedance at its third overtone mode frequency of oscillation. The circuit is as shown below:



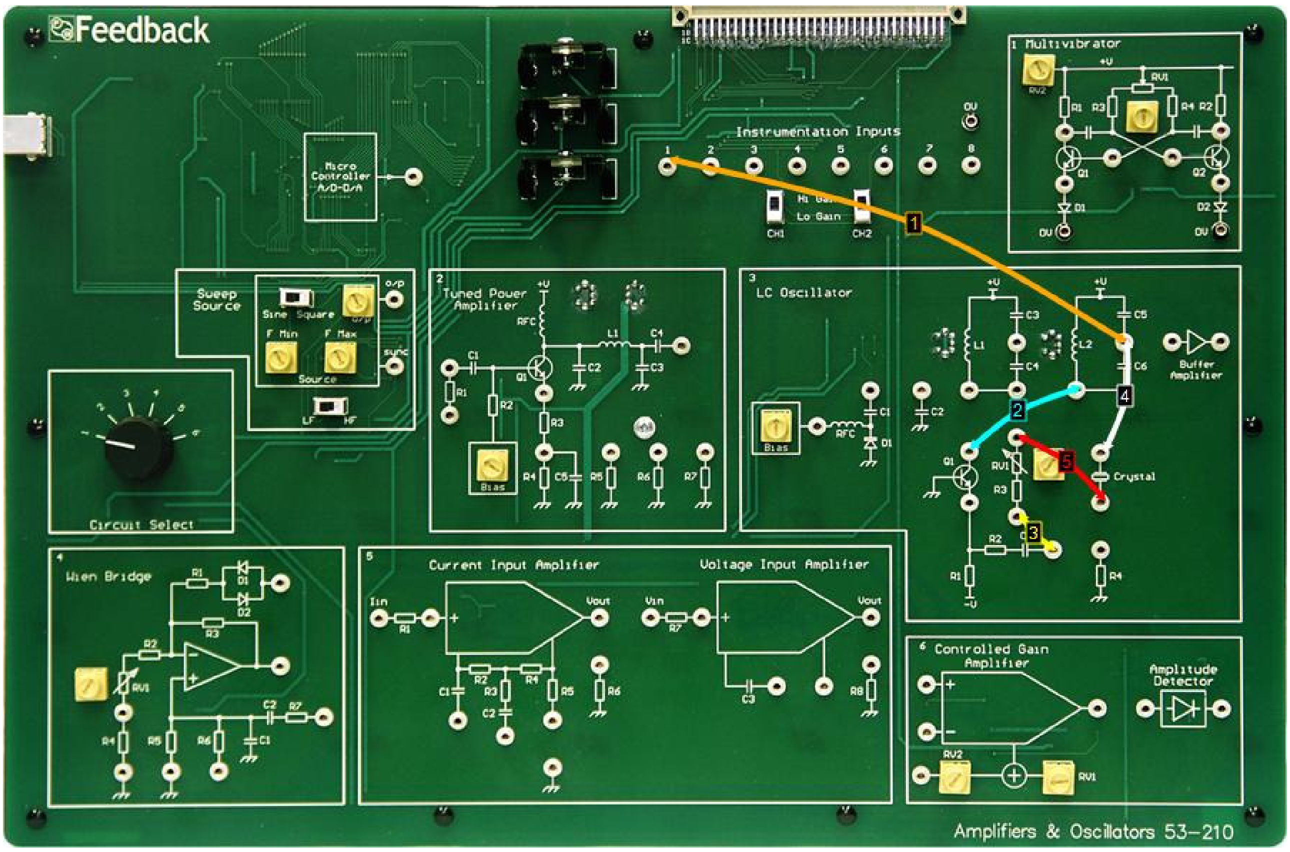
You will adjust the loop gain for sinusoidal oscillation and you will measure the frequency of oscillation.

Practical Note: Crystals are normally designed **either** for fundamental **or** for overtone operation, generally not both. The lowest frequency normally used for overtone operation is about 21MHz, generally much higher. The crystal used in this workboard has been designed for fundamental frequency operation at approximately 1.8MHz (a low frequency). Consequently, it was never intended to operate in its 3rd overtone mode. However, the circuit used can force it to operate in this fashion—albeit in a mode of vibration that is not normally defined—and thus the resulting frequency of operation is quite a way from the theoretical ‘three times’ that is normally expected for 3rd overtone operation. With a much higher frequency device, this 3x ratio is much more closely achieved.

Block Diagram



Make Connections Diagram



Practical 2: Overtone Operation

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that the **Circuit Select** switch, on the left of the workboard, is set to position **3**.

Ensure that the **CH1** switch is set to **Lo Gain**

Set the **RV1** control within the **LC Oscillator** circuit block to half scale.

Open the oscilloscope and adjust the RV1 control until the circuit just starts to oscillate. Note how this position compares with that required for just oscillation at the fundamental frequency. You should see that more feedback is required to get the circuit to oscillate at the overtone frequency.

Open the spectrum analyser and note that the spectrum is essentially just a single frequency, signifying that the output is a sine wave.

Open the counter and note the frequency of the output signal. This is the third overtone frequency of oscillation for the crystal.

Compare this overtone frequency with the fundamental frequency found in Practical 1 and calculate the ratio between them.

Wien Bridge Oscillator

Objectives

To become familiar with the operation of a Wien Bridge phase shift network

To appreciate the requirements for a circuit to oscillate and produce a sinusoidal output signal

To verify that the above requirements are met for the Wien Bridge Oscillator circuit

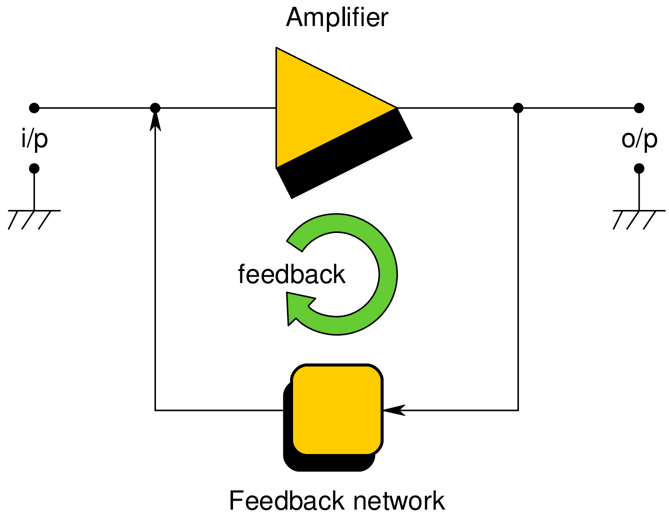
To understand the need for amplitude stabilization for such an oscillator circuit and to investigate a simple method of achieving this

Requirements for Oscillation

An oscillator is a circuit that produces an output signal for no input signal.

An alternative way of thinking of an oscillator is that it is a circuit that converts dc power (which comes from the dc supply to the circuit) into signal power that is available at the output terminals.

Consider an amplifier with feedback applied from the output back to the input, as shown in the following diagram.



The signal fed back can be such that it either helps to cancel or reinforce the original input signal, depending on the **phase** of the feedback signal.

Signals that are fed back in phase with the original input signal will reinforce its effect

Signals that are fed back in anti-phase with the original input signal will reduce its effect

This gives rise to two forms of feedback:

Signals that are fed back in phase with the original input signal give **positive feedback**

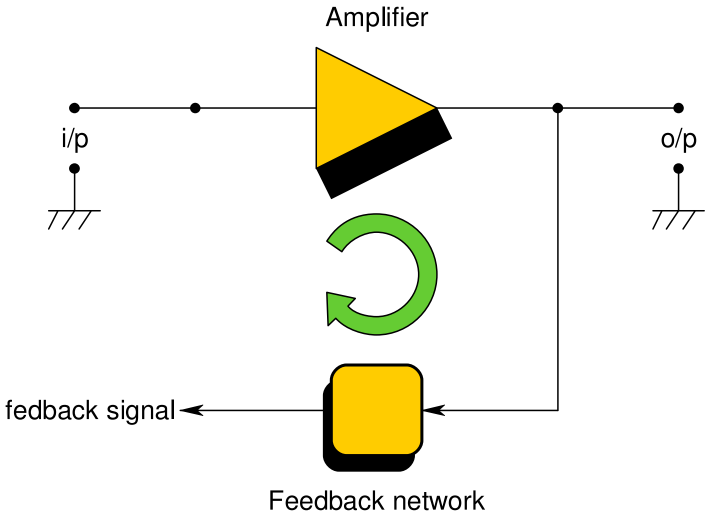
Signals that are fed back in anti-phase with the original input signal give **negative feedback**

An oscillator requires positive feedback to be applied.

The combination of the amplifier and the feedback network is known as the **feedback loop**. For positive feedback to be applied the signal fed back round the loop must be such that it is **in phase** with the original input signal. This means that the **total** phase shift, first through the amplifier and then through the feedback network, must be 0 degrees. This is usually referred to as **the loop phase shift**.

Having determined that loop phase shift must be zero to get full signal reinforcement, the next question is "how much signal needs to be fed back to get sustained oscillation?"

To answer this question consider the circuit above, but with the feedback loop 'broken', as shown in the diagram below.



Suppose the amplifier had a gain of 3 and the feedback network attenuated the signal by a factor of 5 (i.e. it has a 'gain' of 0.2). This means that the feedback signal will be 0.2 of the amplified signal, i.e. 0.2 x 3 times the original input signal: = 0.6 of the input amplitude.

The combination of the amplifier gain and the feedback network attenuation is called the **loop gain**.

To be able to maintain an output, the attenuation of the feedback network must at least be made up by the gain of the amplifier. That **means that the loop gain must be at least unity**.

In this instance, the feedback signal is less than the original input signal so, if the feedback loop were reconnected and the input signal removed, there would not be enough feedback to maintain an output. The loop gain is less than unity. It is usually much easier to change the gain of an amplifier than the attenuation of the feedback network thus, to get unity loop gain, in this instance the amplifier gain must be increased to at least 5.

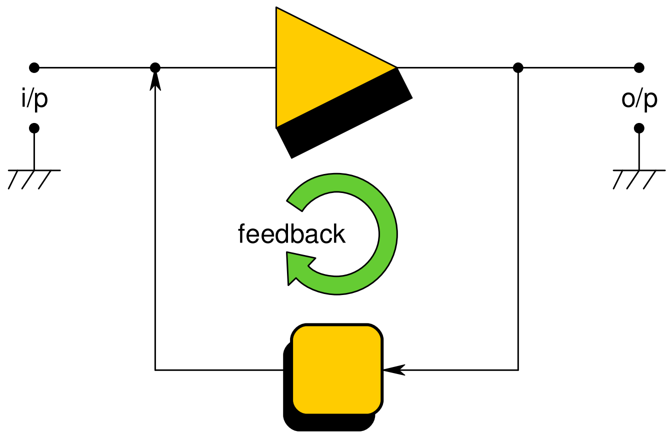
Thus, the criteria for oscillation are:

the loop phase shift must be zero and

the loop gain must be at least one

The Wien Bridge Oscillator

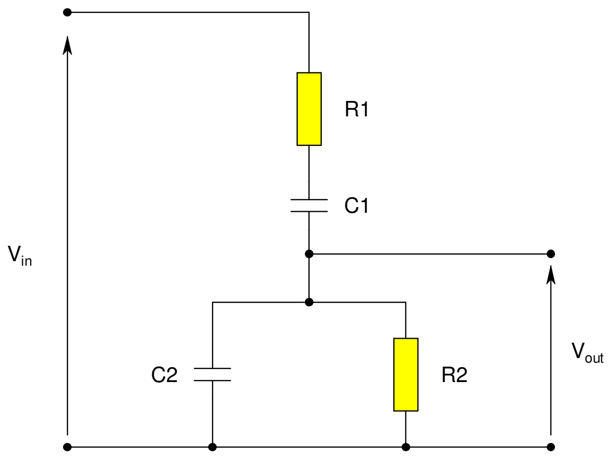
An oscillator comprises an amplifier and a feedback network, as shown in the diagram below.



The feedback must be positive feedback and of sufficient magnitude to maintain an output signal, even when the input is removed.

Thus, the criteria for oscillation are:

the loop phase shift must be zero and
the loop gain must be at least one
A Wien Bridge oscillator uses a feedback network comprising two resistors and two capacitors, as shown below.



The relationship between V_{out} and V_{in} can be found using the normal potential divider equation:

$$\frac{V_{out}}{V_{in}} = \frac{\mathrm{R_2\ in\ parallel\ with\ C_2}}{(\mathrm{R_1\ in\ series\ with\ C_1}) + (\mathrm{R_2\ in\ parallel\ with\ C_2})} = \frac{Z_2}{Z_1 + Z_2}$$

To simplify this it is easier to consider the reciprocal of the expression:

$$\frac{V_{in}}{V_{out}} = \frac{Z_1 + Z_2}{Z_2} = 1 + \frac{Z_1}{Z_2}$$

where,

$$Z_1 = R_1 + \frac{1}{j\omega C_1} \quad \text{and} \quad Z_2 = \frac{R_2}{1 + j\omega C_2 R_2}$$

The equation can be manipulated to give

$$\frac{V_{in}}{V_{out}} = \frac{1 - \omega^2 C_1 C_2 R_1 R_2 + j\omega(C_1 R_1 + C_2 R_2 + C_1 R_2)}{j\omega C_1 R_2}$$

which has zero phase shift where

$$1 - \omega^2 C_1 C_2 R_1 R_2 = 0$$

At this frequency,

$$\frac{V_{in}}{V_{out}} = \frac{C_1 R_1 + C_2 R_2 + C_1 R_2}{C_1 R_2}$$

giving the gain of the network as

$$\frac{V_{out}}{V_{in}} = \frac{C_1 R_2}{C_1 R_1 + C_2 R_2 + C_1 R_2}$$

This has to be made up by the gain of the amplifier.

If R_1 and R_2 (= R) are equal and C_1 and C_2 are also equal (= C), these equations simplify to

$$\omega^2 = \frac{1}{C^2 R^2}$$

giving

$$f = \frac{1}{2\pi CR}$$

as the frequency for zero phase shift and

$$\frac{V_{out}}{V_{in}} = \frac{1}{3}$$

as the gain of the network at this frequency.

Therefore, to get a loop gain of at least unity the amplifier must have a gain of at least 3.

Amplitude Stabilisation of Oscillators

To maintain oscillation the loop gain of an oscillator must be unity, or greater.

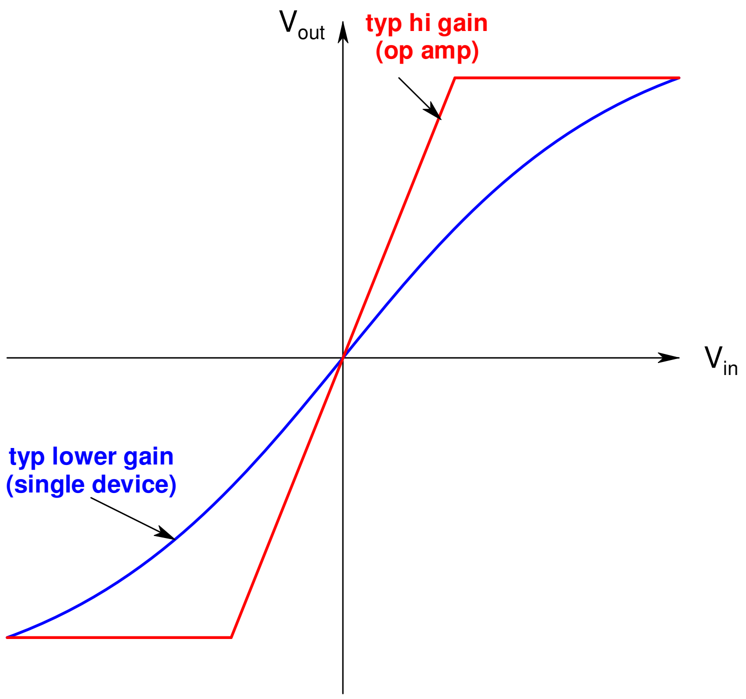
If the loop gain is made exactly unity the circuit would, theoretically, oscillate. However, any variations in the parameters of the feedback and amplifying devices due to such things as ageing, temperature changes, differences between devices in a production run, etc. may easily cause the loop gain to fall to less than one. If that happened, the circuit would cease to oscillate.

Thus, in practice, the gain of the amplifier is made somewhat greater than that required for a loop gain of unity, so that any variations do not bring it lower than one.

Making the loop gain greater than unity means that more signal is being fed back, resulting in a build-up of output amplitude. This happens almost instantaneously at switch-on. However, any practical circuit will only have finite supply voltage rails, so the output amplitude will soon be limited by the operation of the amplifying device(s) approaching these rails.

The diagram below illustrates typically how the characteristics of a high gain device, such as an operational amplifier, and a lower gain single transistor might approach the supply rails (saturation).

Notice that, typically, the op amp has a greater range of linear operation, but that the onset of saturation is very abrupt. In comparison, a single device may typically have a smaller truly linear range, but with the onset of non-linearities being much more gradual.



The onset of non-linearities effectively lowers the gain of the amplifier (and therefore the loop gain) and thus an effect occurs whereby any tendency for the output amplitude to increase (loop gain greater than unity) will immediately result in a lowering of loop gain, thus resulting in a fall in amplitude. A dynamic equilibrium will result and the output amplitude will remain essentially constant.

If the onset of non-linearities is gradual, this amplitude stabilising effect results in little distortion of the output waveform. However, if the onset is abrupt (such as in the case of an op amp) it is extremely difficult to get stabilisation without severe distortion. The output may be almost a square wave.

To combat this distortion, most op amp oscillator circuits include extra circuitry to make the onset of non-linearities less abrupt. The Wien Bridge oscillator circuit on the workboard has back-to-back diodes that can be included in the feedback path to produce this effect, as the diode characteristics ‘round off’ the op amp onset and make the curve more like the single device one.

The advantage of using an op amp is that the gain can be accurately determined by the feedback components and there is far less variation due to different devices, temperature, ageing, etc.

The disadvantage of using op amps comes when the required frequency of operation is above a few hundred kHz, as high-speed op amps are uncommon.

Practical 1: Oscillation Point Analysis

Objectives and Background

In this practical an operational amplifier is provided with a simple negative feedback network that allows its gain to be adjusted within narrow limits.

A second, positive feedback network incorporates the Wien Bridge, comprising series and parallel-connected R and C.

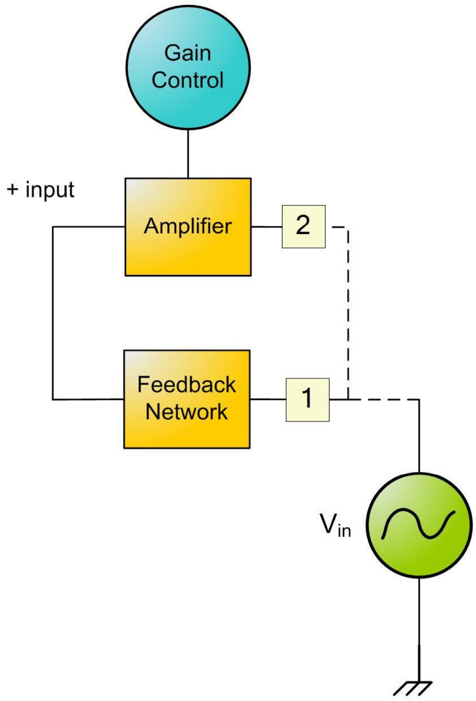
The positive feedback has zero phase-shift at just one frequency. It is shown in the Theory that **both zero phase shift and a particular value of gain are needed to maintain oscillation**. You will use the Gain Phase Analyser to plot the open loop Bode plot to determine the frequency that gives zero phase shift and also the loop gain at this frequency.

You will close the loop and adjust the gain to see that the above criteria are met.

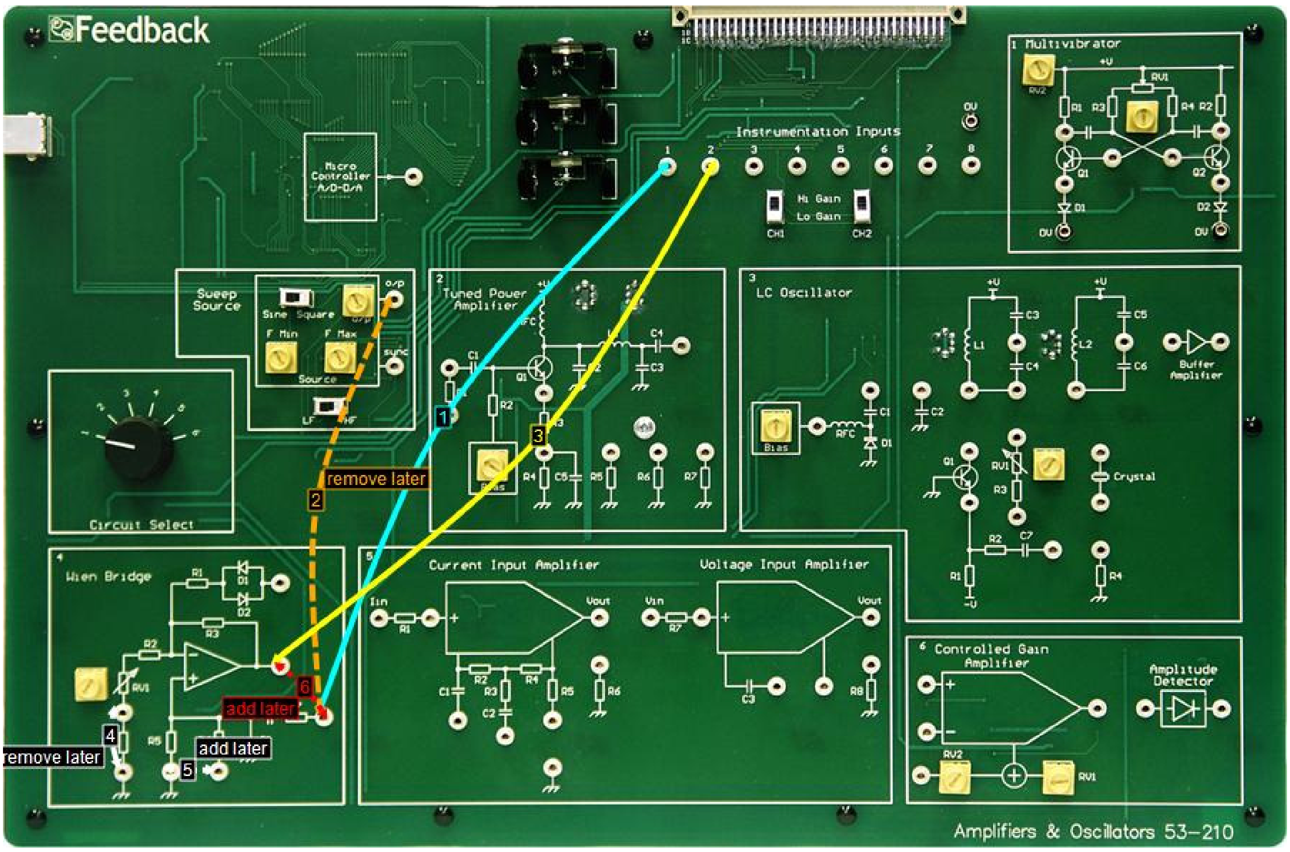
You will see that excessive gain causes oscillation to build up until limited by the available output swing from the amplifier.

You will need to be careful when adjusting the gain control in order to see oscillations building up and dying down.

Block Diagram



Make Connections Diagram



Practical 1: Oscillation Point Analysis

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **Wien Bridge** circuit block, located towards the bottom-left of the workboard.

Ensure that you have the **Circuit Select** switch set to **4**.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 to **Hi Gain**.

Identify the **Sweep Source** circuit block at the left-hand centre of the workboard. In this block, set the **Sine**/**Square** switch to Sine and the **LF**/**HF** switch (just below the Source block) to LF. Set both the **FMin** and **FMax** controls to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 320Hz.

Open the oscilloscope and use the **o**/**p** control on the sweep source to set its output amplitude (yellow probe on monitor point 1) to approximately 0.8V pk-pk. You should also be able to see the output of the amplifier on the blue trace (monitor point 2). At this frequency the gain of the loop is much less than one.

Close the oscilloscope and open the Gain Phase Analyser (GPA).

On the GPA, set FMin to approximately 320Hz and FMax to approximately 10kHz.

Set the **RV1** control in the Wien Bridge circuit to half scale.

Plot the Bode (amplitude and phase) response of the amplifier between these frequencies, noting the variations with frequency. Vary RV1 and note how the gain of the loop varies. You will need to do this slowly, as the GPA takes time to plot at these low frequencies.

Use the cursor to find the frequency at which the loop phase shift is zero. At this frequency, note the loop gain for settings of RV1 of minimum, half scale and maximum.

Note also the position of RV1 that gives a loop gain of exactly unity (0dB).

Switch the GPA to give Nyquist plots and see the form of the plot for loop gains greater and less than unity. See how the loci relate to the 0dB, 0 degree point.

Switch the GPA back to give Bode plots. Now you will see what happens when changes are made in the values of the feedback components.

Ensure that RV1 is adjusted to give a loop gain of just 0dB. Now link R5 and R6, as shown by connection 5 on the Make Connections diagram. This has the effect of halving the value of the resistor in the parallel branch of the Wien Bridge.

Note the change in loop gain and, more importantly, the change in the frequency at which the loop phase shift is zero degrees.

Remove the link that you last made and also remove the link across **R4**, as shown by connection 4 in the Make Connections diagram. Note the change in the loop gain, as shown by the Bode plot., but also note that there is little change in the zero degree phase shift frequency. Changing R4 only changes the gain of the amplifier – not the properties of the Wien Bridge.

The measurements that you have been taking are with the circuit ‘open loop’. Now you will remove the Sweep Source and close the loop and see if the circuit oscillates.

Close the GPA and remove connection 2 and add connection 6, as shown on the Make Connections diagram. This disconnects the source and closes the feedback loop to make the circuit into an oscillator.

Carefully adjust RV1 and note the position that just gives oscillation. Compare this with that found for 0dB loop gain.

Open the counter and measure the frequency of oscillation for the position of RV1 that just gives oscillation. Compare this with the frequency that you measured on the GPA that gave zero degrees loop phase shift.

Note how difficult it is to adjust the gain to get oscillation with a sine wave (undistorted) output. This is because of the very abrupt onset of non-linearities for the op amp. In the next Practical you will introduce the diode network and see the difference it makes.

Practical 2: Amplitude Stabilisation

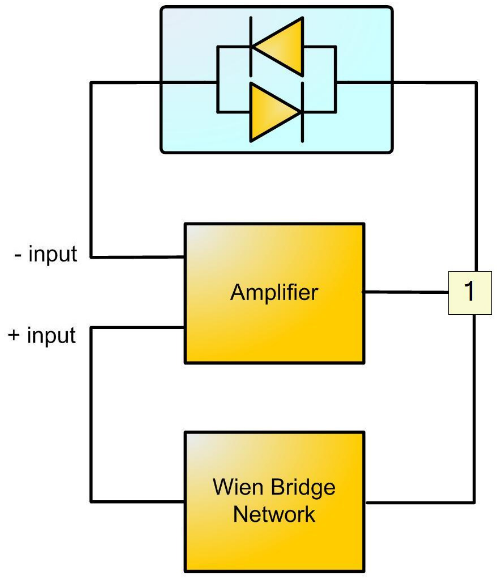
Objectives and Background

In this Practical you will, firstly, try to manually adjust the loop gain of the circuit to achieve oscillation and produce a sinusoidal output that has a stable amplitude. You will find that this is extremely difficult to achieve.

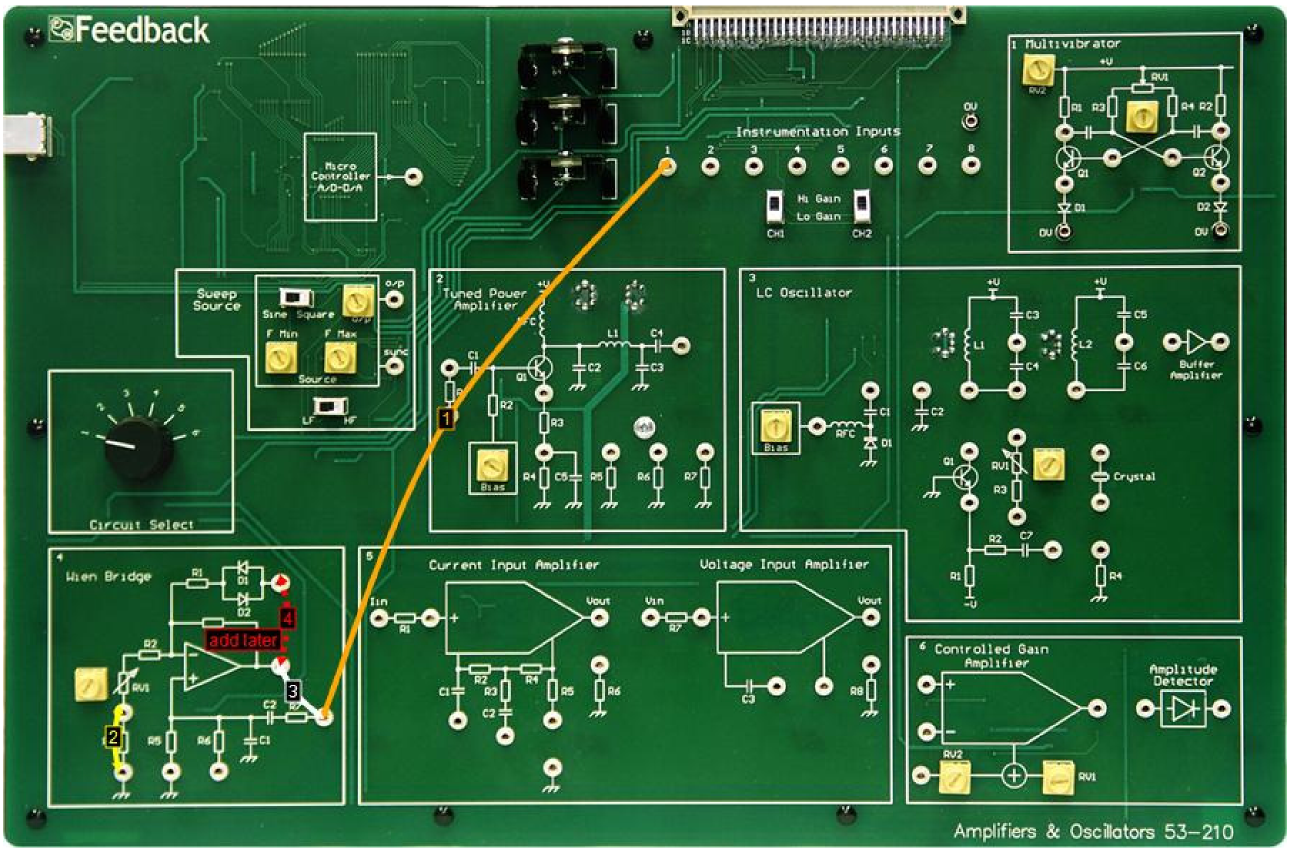
You will then introduce a diode network into the circuit, that uses the non-linearity of the diode characteristics to automatically, and dynamically, adjust the loop gain to unity.

You will see that the modified circuit gives a stable amplitude sinusoidal output signal.

Block Diagram



Make Connections Diagram



Practical 2: Amplitude Stabilisation

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that you have the **Circuit Select** switch set to **4**.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 to **Hi Gain**.

Set **RV1** within the **Wien Bridge** circuit block to half scale.

Open the oscilloscope. You should see that the circuit is oscillating, but that the output is limiting (square wave).

Adjust the loop gain using RV1 and verify that it is almost impossible to get the oscillator to produce a stable amplitude sine wave output.

Add the diode network by making connection 4, as shown on the Make Connections diagram.

Observe the change in the output waveform and adjust RV1 to see its effect. Note that it is now possible to get a sine wave output from the circuit and that its amplitude is much more easily controlled.

Power Amplifier (1)

Objectives

To become familiar with the operation of a tuned power amplifier

To investigate the frequency selectivity of such an amplifier

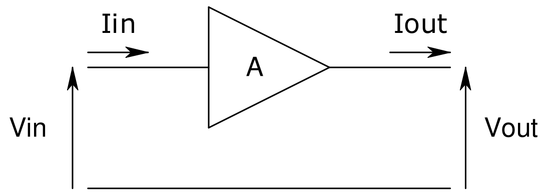
To determine the input resistance and the power gain of the amplifier

To investigate the effect of changing load resistance on the output power of such an amplifier

Amplifiers

Gain

The block diagram of a general amplifier is given below.



The gain of the amplifier is denoted by the symbol A. The definition of A is given by:

$$\text{Gain, } A = \frac{(output\ power)}{(input\ power)}$$

This is called the **power gain** of the amplifier.

If a circuit does not have power gain, then it is not an amplifier!

As you can see, there is present input current and input voltage and, at the output, the corresponding output current and voltage. As power is the product of current and voltage, this gives the expression for the gain:

$$A = \frac{Vout.Iout}{Vin.Iin}$$

Now:

$\frac{Vout}{Vin} = A_V$, the **voltage gain** of the amplifier

and:

$\frac{Iout}{Iin} = A_I$, the **current gain** of the amplifier

Therefore:

$$A = A_V \,.\, A_I$$

Note: that it is quite possible for an amplifier to have a voltage gain of less than one; however, its corresponding current gain must be high enough to give a power gain greater

than unity for it to be classed as an amplifier. This also works the other way: if the current gain is less than one, the voltage gain must be high enough.

Going back to the expression for gain of:

$$A = \frac{Vout.Iout}{Vin.Iin}$$

This can be further re-arranged by defining two more relationships:

$\frac{Vout}{Iin} = G_m$, the **transresistance gain** of the amplifier

and:

$\frac{Iout}{Vin} = R_m$, the **transconductance gain** of the amplifier

Giving, therefore:

$$A = G_m \, . \, R_m$$

Using a similar argument to before, to qualify as an amplifier, it is quite possible for either one or the other of these terms to be less than one providing the product of the two is greater than unity.

Types of Amplifier

The input signal to an amplifier may be a current or it may be a voltage. Therefore, this gives rise to two types of amplifier: the current input amplifier and the voltage input amplifier.

Each of these types of amplifier may be further sub-divided, as each can give a current output or a voltage output.

There are, therefore, four general forms of amplifier.

Different applications require amplifiers to have different properties of amplification.

An example

Consider a fibre optic communications system with a great distance between transmitter and receiver. Because of the attenuation of light along the length of fibre it is often necessary to compensate for this loss by having ‘repeater’ circuits at intervals along the fibre. The purpose of these circuits is to detect the incoming light signal, convert it into an electrical signal that can be amplified and then used to drive a secondary light source that

provides a regenerated signal for transmission further down the cable.

A typical sensor for the input of such a repeater is a photodiode. This will produce a current that is proportional to the light intensity. A typical output device might be a light-emitting diode, or perhaps a laser diode, which also needs a current signal to drive it. The amplifier within the repeater must thus take the small current output from the photodiode and amplify it to drive the LED or laser.

Therefore, the amplifier required has to be a **current amplifier**. The requirements for such a circuit are:

Its input should affect the signal current from the photodiode as little as possible,

It should have output circuitry that maximises the current transfer out of the amplifier,

It should have current gain.

To achieve this, it should have **as low an input resistance as possible** and its output should look like an ideal current source (i.e. it should have **as high an output resistance as possible**).

Another example

Consider a communications receiver system. Within the receiver there are circuits that perform functions such as high frequency amplification of the signals, the selection of the required signal (filters) and frequency translation of the signal to a lower frequency (mixers). Because of current limitations on the operating speed of analogue-to-digital converters and other digital circuitry, these functions are normally performed by analogue circuits. However, it is common practice to convert high frequency signals to a much lower frequency (often a few tens of kHz) so that such functions as demodulation or decoding and the final signal processing can be done using digital techniques. DSP (digital signal processing) circuits are widely used to do this.

Typically, DSP chips require input voltage signals of a few volts amplitude. The output signal from the analogue part of the receiver system may be only tens of millivolts in amplitude. An amplifier is therefore required to ‘bridge this gap’.

Therefore, this amplifier has to be a **voltage amplifier**. The requirements for such a circuit are:

Its input should affect the signal voltage from the analogue circuitry as little as possible,

It should have output circuitry that maximises the voltage transfer out of the amplifier,

It should have voltage gain.

To achieve this, it should have **as high an input resistance as possible** and its output should look like an ideal voltage source (i.e. it should have **as low an output resistance as possible**).

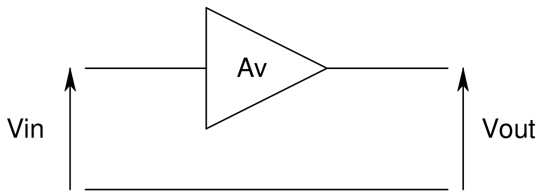
You can imagine that the requirements for the two example amplifiers above will result in completely different circuitry to satisfy them.

The two other types of amplifier also find uses in electronics and communications systems. For example, a field effect transistor amplifier is an example of a **transconductance amplifier** and **transresistance amplifiers** are widely used in audio

mixers and digital-to-analogue converters.

The Voltage Amplifier

A **voltage amplifier** is one to which an input voltage is applied and an output voltage results. The block diagram for such a system is given below.



The **voltage gain** of such an amplifier is given by

$$\text{Voltage gain,} \quad A_v = \frac{V_{out}}{V_{in}}$$

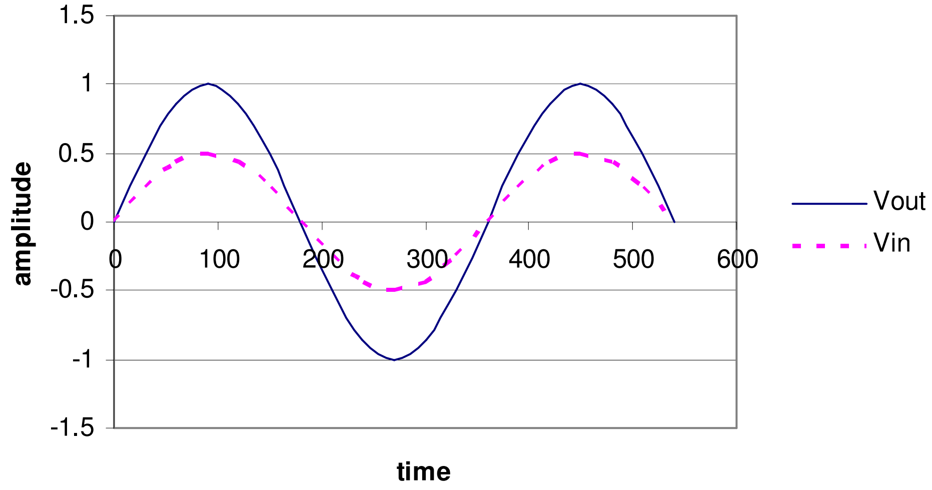
Ideally, connecting a voltage amplifier into a circuit should not affect the input voltage in any way. To achieve this, the input impedance of the amplifier should be as high as possible.

Also, ideally, connecting a load onto the output of the amplifier should not change the output voltage in any way. For this to happen, the output impedance of the amplifier should be as low as possible.

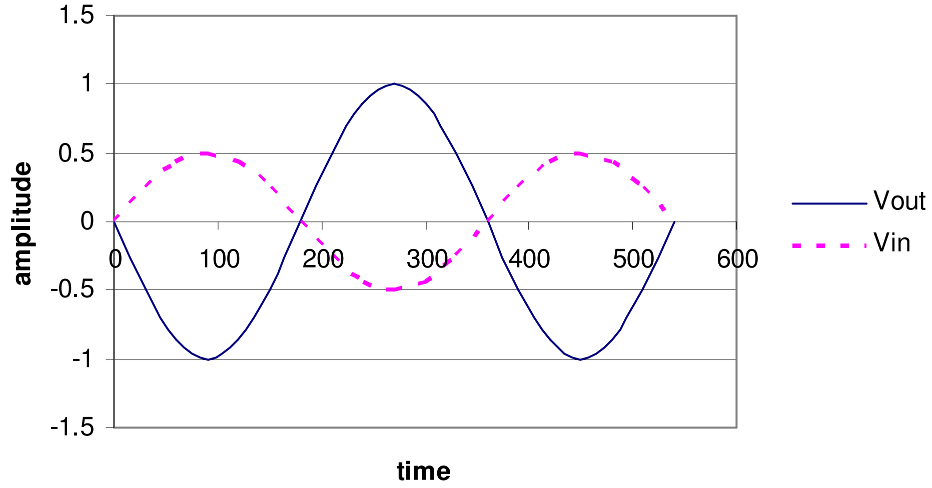
The ideal properties for a voltage amplifier are thus:

Voltage gain	Yes
Input impedance	Very high
Output impedance	Very low

The output voltage may follow the input voltage directly, or it may be inverted in polarity. This is illustrated below:



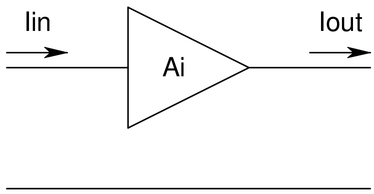
This shows the output following the input directly. The output is said to be **in phase** with the input. Another name for such an amplifier is a **non-inverting amplifier**. Note, also, that the output is amplified with respect to the input.



This shows the output inverted with respect to the input. The output is said to be in **anti-phase** with the input. Another name for such an amplifier is an **inverting amplifier**. Note, also, that the output is amplified with respect to the input.

The Current Amplifier

A **current amplifier** is one to which an input current is applied and an output current results. The block diagram for such a system is given below.



The **current gain** of such an amplifier is given by

$$\text{Current gain,} \quad A_i = \frac{I_{out}}{I_{in}}$$

Ideally, connecting a current amplifier into a circuit should not affect the input current in any way. To achieve this, the input impedance of the amplifier should be as low as possible.

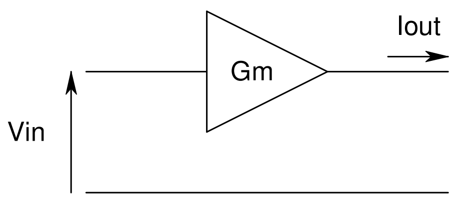
Also, ideally, connecting a load onto the output of the amplifier should not change the output current in any way. For this to happen, the output impedance of the amplifier should be as high as possible.

The ideal properties for a voltage amplifier are thus:

Current gain	Yes
Input impedance	Very low
Output impedance	Very high

The Transconductance Amplifier

A **transconductance amplifier** is one to which an input voltage is applied and an output current results. The block diagram for such a system is given below.



The **transconductance gain** of such an amplifier is given by

$$\text{Transconductance gain,} \quad G_m = \frac{I_{out}}{V_{in}}$$

Ideally, connecting a voltage amplifier into a circuit should not affect the input voltage in any way. To achieve this, the input impedance of the amplifier should be as high as possible.

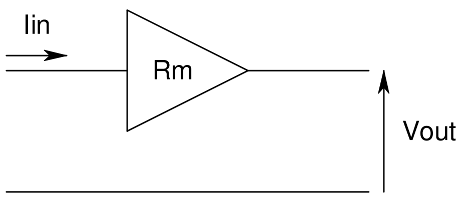
Also, ideally, connecting a load onto the output of the amplifier should not change the output current in any way. For this to happen, the output impedance of the amplifier should be as high as possible.

The ideal properties for a transconductance amplifier are thus:

Transconductance gain	Yes
Input impedance	Very high
Output impedance	Very high

The Transresistance Amplifier

A **transresistance amplifier** is one to which an input current is applied and an output voltage results. The block diagram for such a system is given below.



The **transresistance gain** of such an amplifier is given by

$$\text{Transresistance gain,} \quad R_m = \frac{V_{out}}{I_{in}}$$

Ideally, connecting a transresistance amplifier into a circuit should not affect the input current in any way. To achieve this, the input impedance of the amplifier should be as low as possible.

Also, ideally, connecting a load onto the output of the amplifier should not change the output voltage in any way. For this to happen, the output impedance of the amplifier should be as low as possible.

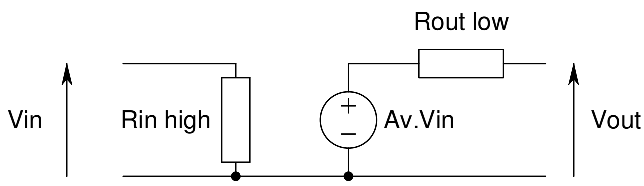
The ideal properties for a transresistance amplifier are thus:

Transresistance gain	Yes
Input impedance	Very low
Output impedance	Very low

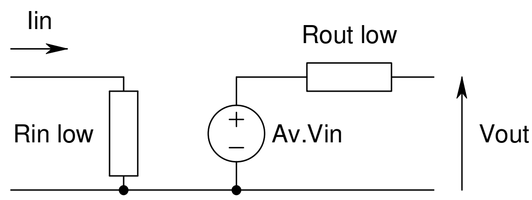
Summary of Amplifier Properties

A summary of the properties of the four types of amplifier, together with an equivalent circuit for each, is given below.

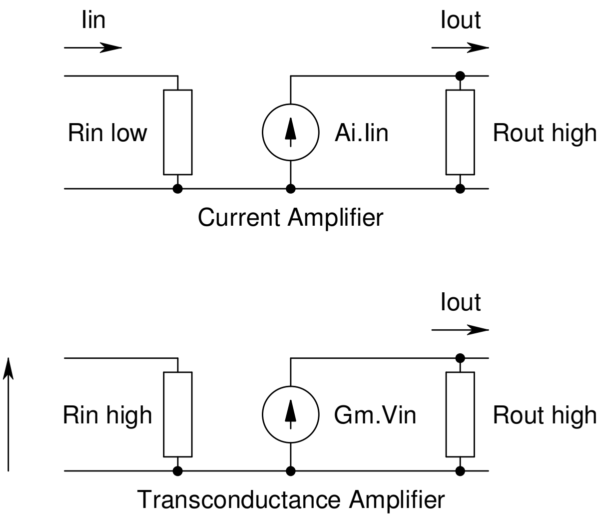
Amplifier Type	**Gain Required**	**Input Resistance**	**Output Resistance**
Voltage Amplifier	Voltage A_V	Very high	Very low
Current Amplifier	Current A_I	Very low	Very high
Transconductance Amplifier	Transconductance G_m	Very high	Very high
Transresistance Amplifier	Transresistance R_m	Very low	Very low



Voltage Amplifier



Transresistance Amplifier


Iin
Iout
Rin low
Ai.Iin
Rout high
Current Amplifier
Iout
Rin high
Gm.Vin
Rout high
Transconductance Amplifier

Classes of Amplifier Operation

RF amplifiers are classified A, AB, B or C according to the phase-angle **(***number of degrees of current flow during each 360-degree RF cycle***)** over which anode- or collector-current flows.

Class A Amplifiers

Class A amplifiers operate over a relatively small portion of a transistor's collector-current (or a vacuum tube's anode-current) range and have continuous collector-current flow throughout each RF cycle. Their efficiency in converting DC-source-power to RF-output-power is poor. DC source power that is not converted to radio frequency output power is dissipated as heat. However, Class A amplifiers do have greater input-to-output waveform linearity (*lower output-signal distortion*) than any other amplifier class. They are most commonly used in small-signal applications where linearity is more important than power efficiency, but also are sometimes used in large-signal applications where the need for extraordinarily high linearity outweighs cost and heat disadvantages associated with poor power efficiency.

Class B Amplifiers

Class B amplifiers have their transistor bases biased near collector-current cut-off, causing collector-current to flow only during approximately 180degrees of each RF cycle. That causes the DC-source-power to RF-output-power efficiency to be much higher than with ClassA amplifiers, but at the cost of severe output cycle waveform distortion. That waveform distortion is greatly reduced in practical designs by using relatively high-Q resonant output circuits to reconstruct full RF cycles.

The effect is the same in principle as pushing a child in a swing through half-swing-cycles and letting the natural oscillatory characteristics of the swing move the child through the other half-cycles. However, low sine-wave distortion results in either case only if the Q of the oscillatory circuit (*the output circuit or the swing*) is sufficiently high. Unless the Q is infinite, which it never can be, the amplitude of one-half cycle will be larger than the other, which is another way of saying there always will be some amount of harmonic energy. (*Coupling an antenna system too tightly to the resonant output circuit of an amplifier will lower its Q, increasing the percentage of harmonic content in the output.*)

Another effective method commonly used to greatly reduce Class B RF amplifier output waveform distortion (*harmonic content*) is to employ two amplifiers operating in "push-pull" such that one conducts on half-cycles where the other is in collector current cut-off. Oscillatory tank circuits are still used in the outputs of Class B push-pull amplifiers to smooth switching transitions from the conduction of one amplifier to the other, and to correct other nonlinearities, but lower-Q circuits can be used for given percentages of harmonic content in the output. (*Output circuits can be loaded more-heavily for given percentages of harmonic output where two amplifiers operate in push-pull.*)

Class AB Amplifiers

As the designation suggests, Class AB amplifiers are compromises between Class A and Class B operation. They are biased so collector current flows less than 360 degrees, but more than 180 degrees, of each RF cycle. Any bias point between those limits can be used, which provides a continuous selection range extending from low distortion, low efficiency on one end to higher distortion, higher efficiency on the other.

Class AB amplifiers are widely used in SSB linear amplifier applications where low distortion and high power-efficiency tend to both be very important. Push-pull Class AB amplifiers are especially attractive in SSB linear amplifier applications because the greater linearity resulting from having one amplifier or the other always conducting makes it possible to bias push-pull Class AB amplifiers closer to the Class B end of the AB scale where the power-efficiency is higher. Alternatively, push-pull Class AB amplifiers can be biased far enough toward the highly linear Class A end of the scale to make broadband operation without resonant output circuits possible in applications where broadband operation or freedom from tuning is more important than power-efficiency.

Class C Amplifiers

Class C amplifiers are biased well beyond cut-off, so that collector current flows less than 180 degrees of each RF cycle. That provides even higher power-efficiency than Class B operation, but with the penalty of even higher input-to-output nonlinearity, making use of relatively high-Q resonant output circuits to restore complete RF sine-wave cycles essential. High amplifying nonlinearity makes them unsuitable to amplify AM, DSB, or SSB signals.

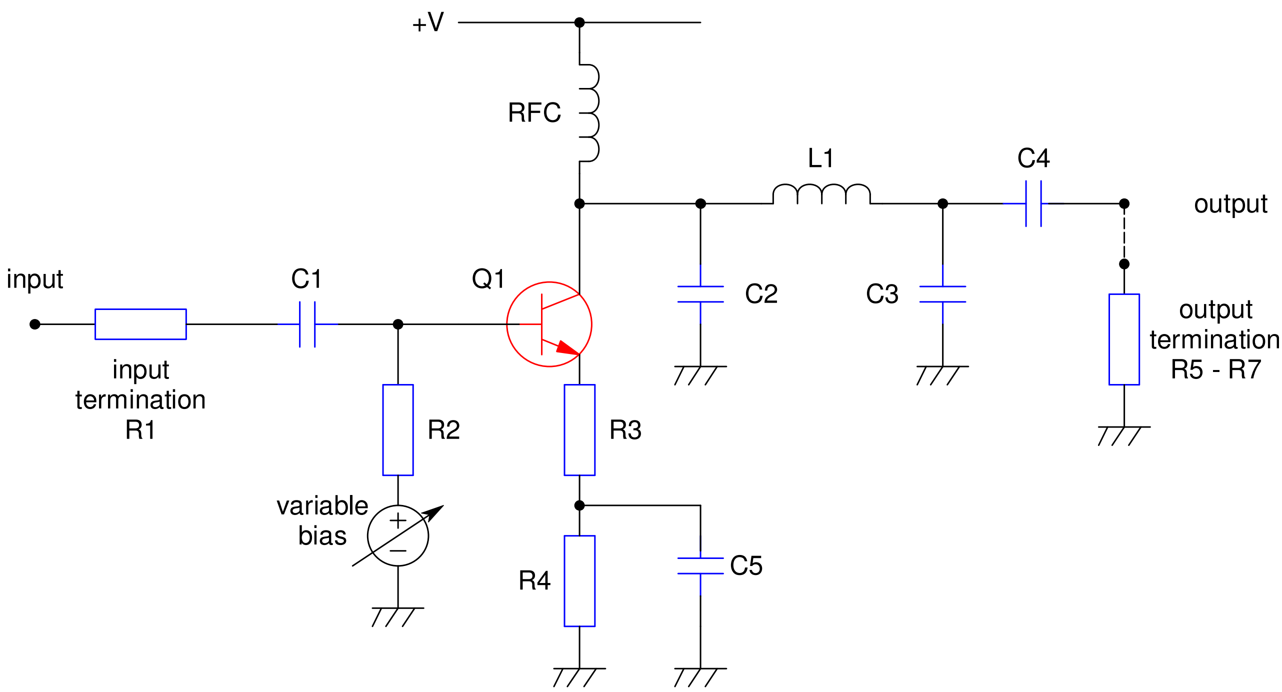
However, most Class C amplifiers can be amplitude modulated with acceptably low distortion by varying the collector voltage, because they generally are operated in the region of collector saturation so that the RF output voltage is very closely dependent upon instantaneous DC collector voltage. They also are commonly used in CW and frequency-shift-keyed radiotelegraph applications and in phase- and frequency-modulated transmitter applications where signal amplitudes remain constant.

Practical 1: Tuned Power Amplifier

Objectives and Background

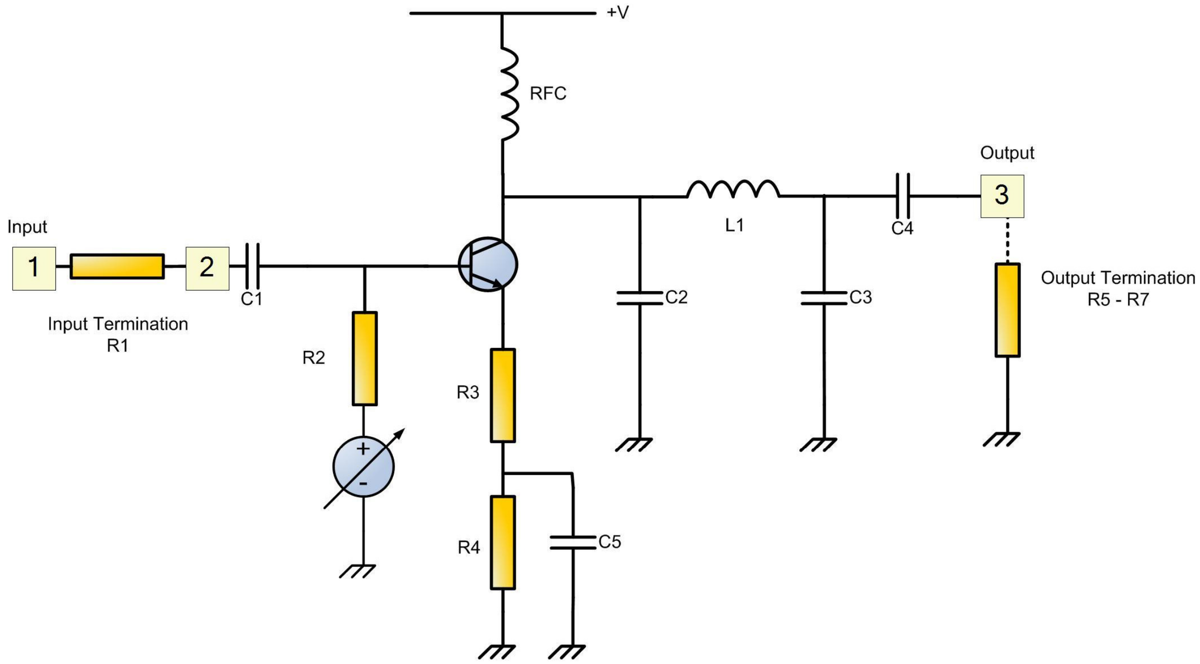
The Power Amplifier that you will be using in this Assignment is based around a bipolar transistor that has a ‘pi’ network of inductor and capacitors in its output (collector) circuit. This network both tunes the amplifier and provides some matching between the transistor and the load.

The circuit for the power amplifier is given below.

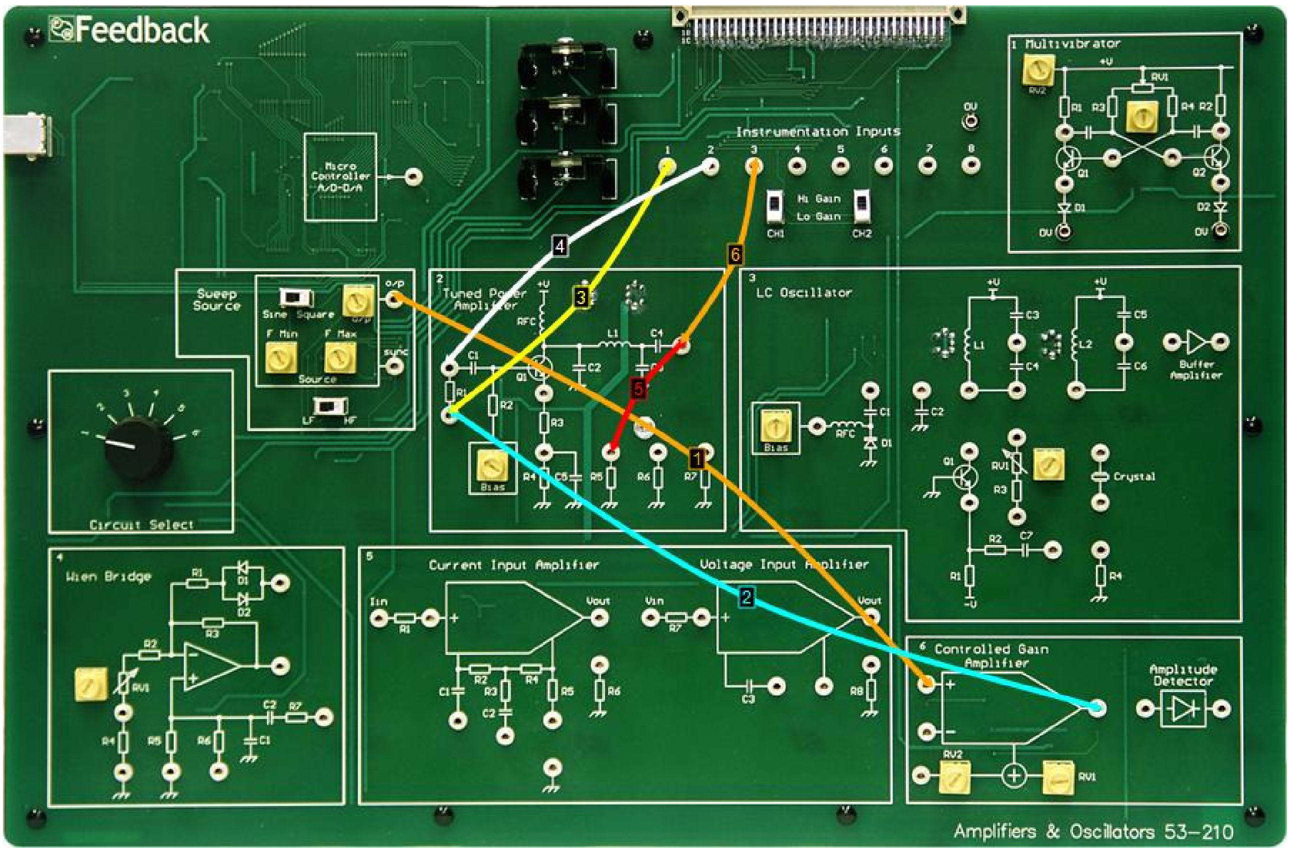


In this Practical you will ascertain that the amplifier does have a frequency selective response and that its frequency oFMaximum output is approximately 1MHz.

Block Diagram



Make Connections Diagram



Practical 1: Tuned Power Amplifier

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **Tuned Power Amplifier** circuit block, located towards the centre of the workboard. Set the **Bias** control in this circuit block to its 9 o'clock position.

Ensure that the **Circuit Select** switch, on the left of the workboard, is set to position **2**.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 switches to **Lo Gain**.

Identify the **Sweep Source** circuit block at the left-hand centre of the workboard. In this block, set the **Sine**/**Square** switch to Sine and the **LF**/**HF** switch to HF. Set both the **FMin** and **FMax** controls to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Identify the **Controlled Gain Amplifier** circuit block. This is to be found in the bottom, right-hand corner of the workboard. Set **RV1** in this block to its maximum (fully clockwise) position and **RV2** to half scale.

Open the oscilloscope and use the **o**/**p** control on the Sweep Source to set the input signal amplitude to the power amplifier to approximately 4V pk-pk.

Open the frequency counter and monitor the frequency of the amplified signal. Use the FMin control on the Sweep Source to vary the frequency of the source and see how the input and output signal amplitudes vary.

You should find that the input amplitude stays relatively constant but that the output amplitude peaks at one particular frequency. This verifies that the amplifier is tuned, i.e. frequency selective. Note the frequency of this peak.

Carefully set the frequency to this peak. It should be very sharp.

Open the phasescope. Carefully adjust the source frequency about the peak to give a phase as close as possible to zero. This is the resonant frequency of the amplifier.

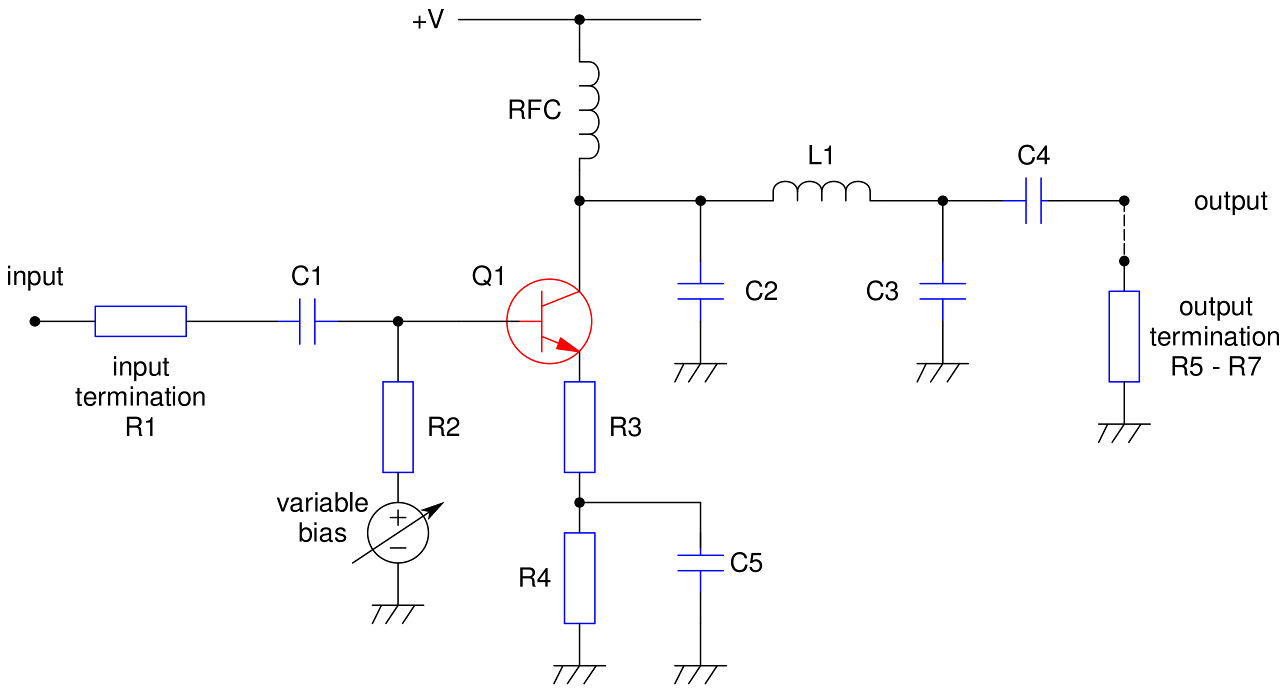
Note that, around the resonant frequency, the phase changes much more rapidly than the amplitude.

Practical 2: Amplifier Input Resistance

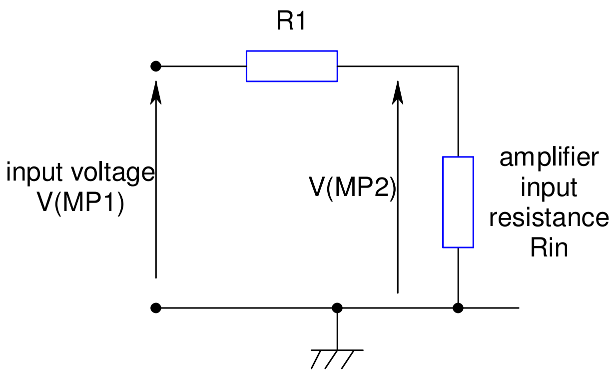
Objectives and Background

In this Practical you will determine the input resistance (R_{in}) of the amplifier.

The circuit of the amplifier is shown below.



At the input of the amplifier, the signal voltage drop across resistor R1 can be used to determine the input resistance of the amplifier. The input circuit is equivalent to:



If the voltages at monitor points 1 and 2 are measured (V(MP1) and V(MP2)) then the input resistance of the amplifier can be determined using potential divider theory:

$$\frac{V(MP2)}{V(MP1)} = \frac{R_{in}}{R_1 + R_{in}}$$

Giving:

$$R_{in} = \frac{V(MP2)}{V(MP1) - V(MP2)} R_1$$

The value of R1 on your workboard is 470Ω.

Because the formulae above contain ratios of voltages, either rms or pk-pk values may be used to get the correct result - providing that you don't have a mixture of both.

You will also determine the power gain for the amplifier (under the conditions of the bias setting used). The formula for power gain is:

$$PowerGain = \frac{P_{out}}{P_{in}}$$

The input power is given by:

$$P_{in} = \frac{V_{in}^2}{R_{in}}$$

and the output power by:

$$P_{out} = \frac{V_{out}^2}{R_L}$$

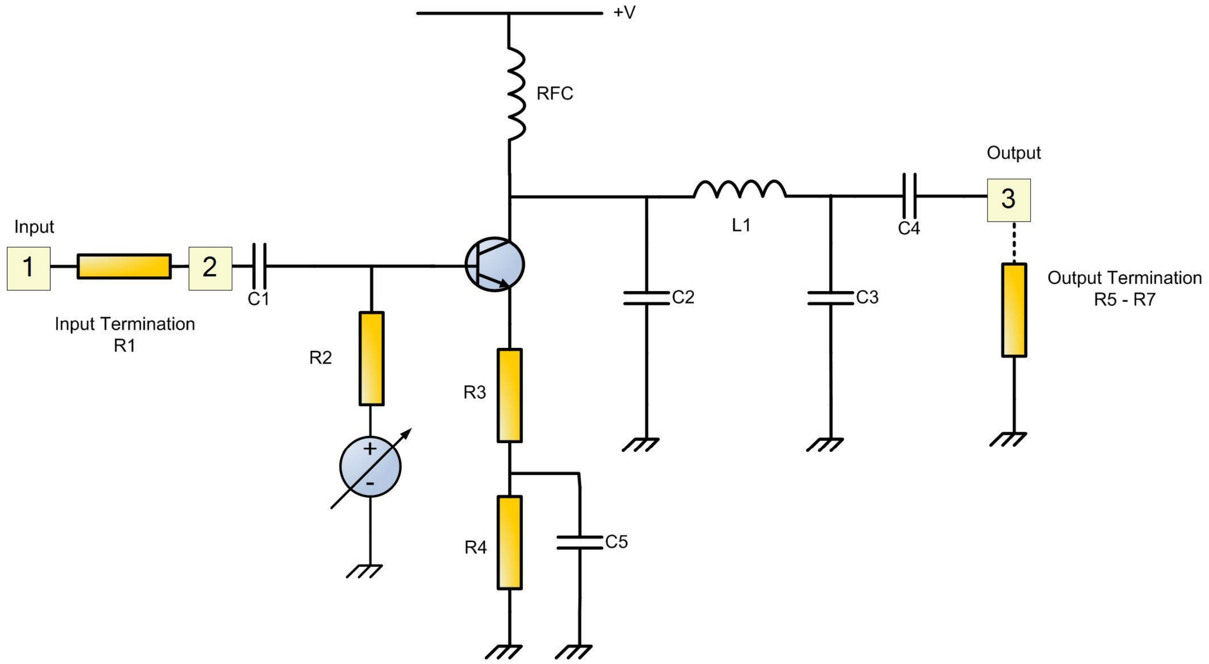
where R_L is the load resistance into which the amplifier is working. In this case, it is 47Ω.

You will calculate the power gain in decibels, using

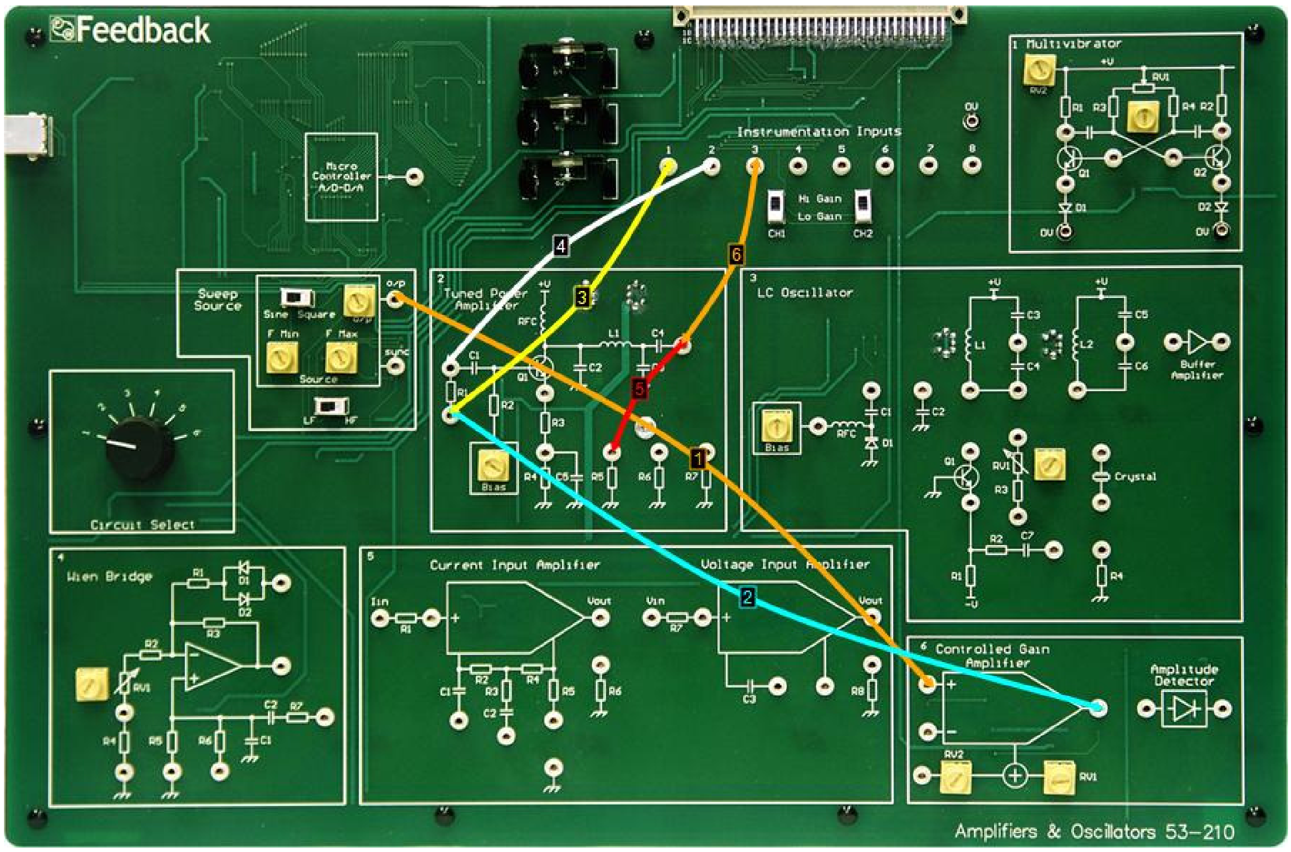
$$PowerGain = 10\log_{10}\frac{P_{out}}{P_{in}} \text{ dB}$$

For the gain calculations, you **must** use rms voltage values to achieve the correct answers.

Block Diagram



Make Connections Diagram



Practical 2: Amplifier Input Resistance

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware (the connections are the same as used in Practical 1).

You can use the same initial control and switch settings as you had when you finished Practical 1.

Open the oscilloscope, the phasescope and the frequency counter.

Use the **o/p** control on the Sweep Source to set the input to the amplifier at monitor point 1 (yellow probe) to be approximately 4V pk-pk.

Use the **FMin** control to set the frequency of the source to the resonant frequency of the amplifier (as shown by zero degrees on the phasescope). This should be approximately 1MHz.

Open the voltmeter and use it to measure the pk-pk input voltage at monitor point 1.

Move the yellow probe to monitor point 2 and measure the pk-pk voltage at this point.

Calculate the input resistance of the amplifier, using the formula given in the background to this Practical.

Now to find the power gain of the amplifier. Remember, rms voltage values must be used in these power calculations (divide the pk-pk value by 2 and then multiply by 0.707).

Using the value that you found for R_{in}, calculate the input power to the amplifier, using the voltage at monitor point 2 as V_{in}.

Measure the output voltage of the amplifier at monitor point 3 and use this to calculate the power into the load.

Divide the output power by the input power and find the gain, both as a ratio and in dB.

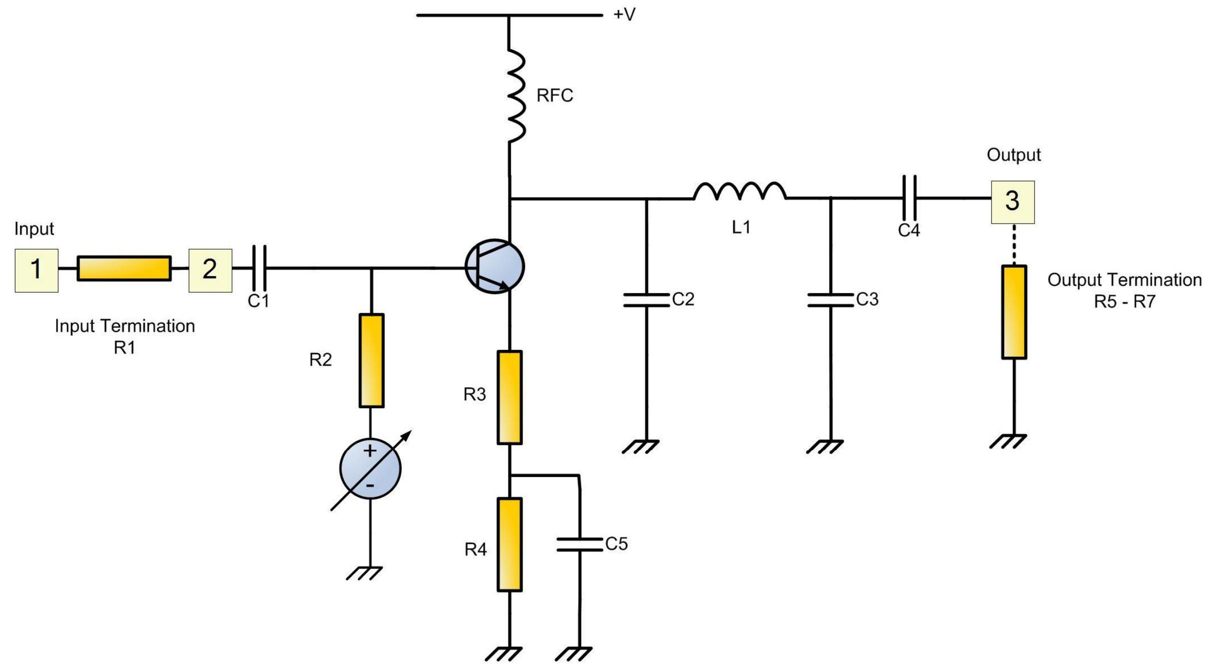
Practical 3: Different Load Resistances

Objectives and Background

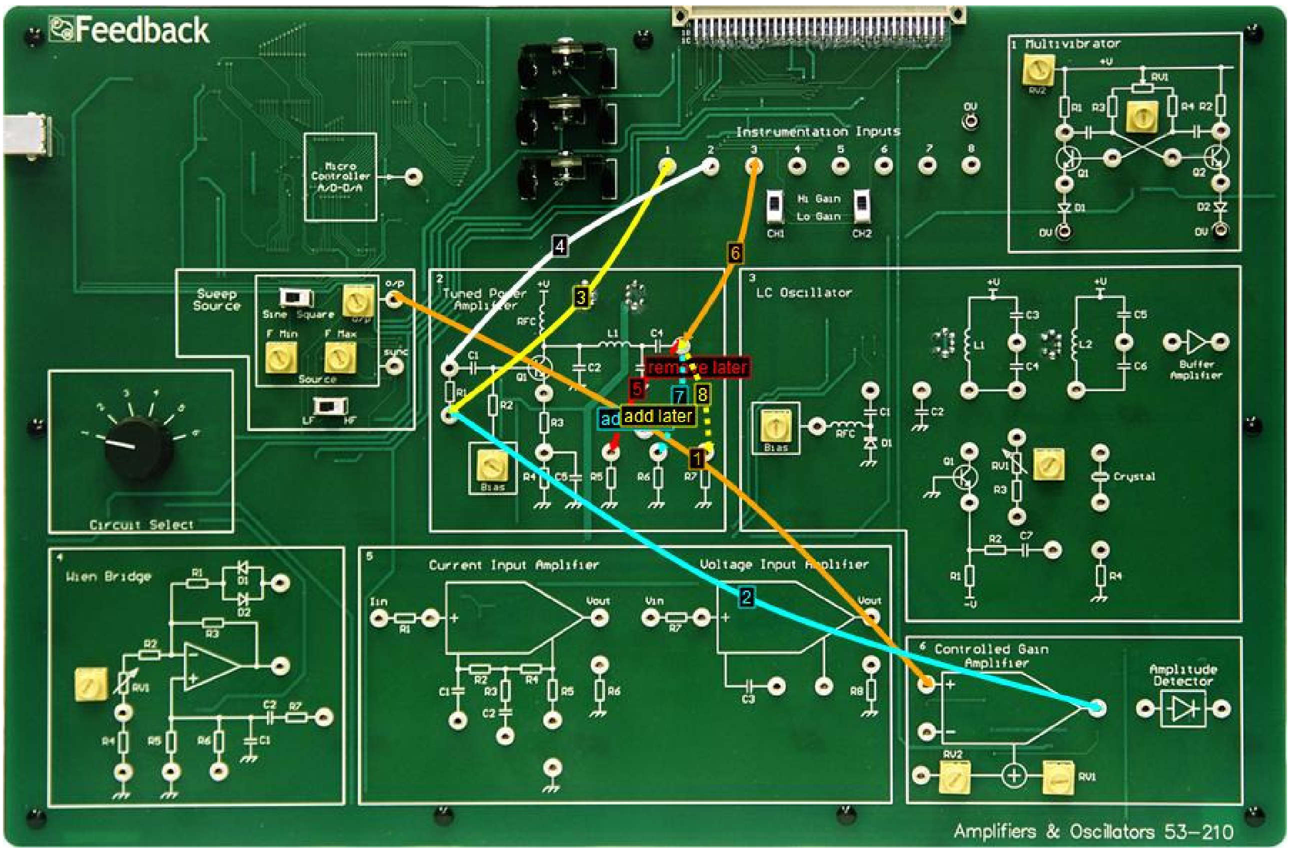
In this Practical you will investigate the performance of the amplifier with different values of load resistance connected to the output.

The 'design' value of load for the amplifier is approximately 50Ω. You will calculate the power output into this load (actually 47Ω) and into load resistances of 100Ω and 10Ω.

Block Diagram



Make Connections Diagram



Practical 3: Different Load Resistances

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware (the connections are the same as used in Practical 2).

You can use the same initial control and switch settings as you had when you finished Practical 2.

Open the oscilloscope and the frequency counter.

Set the **o/p** control on the **Sweep Source** to its maximum (fully clockwise) position.

Use the **FMin** control to ensure that the frequency of the source is set to the resonant frequency of the amplifier.

Open the voltmeter and use it to measure the pk-pk output voltage at monitor point 3.

Calculate the output power into the load. The value of R5 is 47Ω.

Move the load connection to load resistor **R6**, as shown by connection 7 on the Make Connections diagram.

Measure the new output voltage and calculate the output power into this load. The value of R6 is 100Ω.

Move the load connection to load resistor **R7**, as shown by connection 8 on the Make Connections diagram.

Measure the new output voltage and calculate the output power into this load. The value of R7 is 10Ω.

Compare the output powers into the three values of load.

Power Amplifier (2)

Objectives

To appreciate the differences in performance for different classes of operation for a tuned power amplifier

To investigate the change in gain with change in bias applied to the amplifier (change in class of operation)

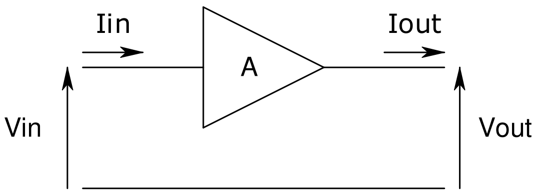
To understand the concept of efficiency as applied to a tuned power amplifier

To investigate the change in efficiency with change in bias applied to the amplifier (change in class of operation)

Amplifiers

Gain

The block diagram of a general amplifier is given below.



The gain of the amplifier is denoted by the symbol A. The definition of A is given by:

$$\text{Gain, A} = \frac{(output\ power)}{(input\ power)}$$

This is called the **power gain** of the amplifier.

If a circuit does not have power gain, then it is not an amplifier!

As you can see, there is present input current and input voltage and, at the output, the corresponding output current and voltage. As power is the product of current and voltage, this gives the expression for the gain:

$$A = \frac{Vout.Iout}{Vin.Iin}$$

Now:

$$\frac{Vout}{Vin} = A_V$$, the **voltage gain** of the amplifier

and:

$$\frac{Iout}{Iin} = A_I$$, the **current gain** of the amplifier

Therefore:

$$A = A_V \, . \, A_I$$

Note: that it is quite possible for an amplifier to have a voltage gain of less than one; however, its corresponding current gain must be high enough to give a power gain greater

than unity for it to be classed as an amplifier. This also works the other way: if the current gain is less than one, the voltage gain must be high enough.

Going back to the expression for gain of:

$$A = \frac{Vout.Iout}{Vin.Iin}$$

This can be further re-arranged by defining two more relationships:

$\frac{Vout}{Iin} = G_m$, the **transresistance gain** of the amplifier

and:

$\frac{Iout}{Vin} = R_m$, the **transconductance gain** of the amplifier

Giving, therefore:

$$A = G_m \,.\, R_m$$

Using a similar argument to before, to qualify as an amplifier, it is quite possible for either one or the other of these terms to be less than one providing the product of the two is greater than unity.

Types of Amplifier

The input signal to an amplifier may be a current or it may be a voltage. Therefore, this gives rise to two types of amplifier: the current input amplifier and the voltage input amplifier.

Each of these types of amplifier may be further sub-divided, as each can give a current output or a voltage output.

There are, therefore, four general forms of amplifier.

Different applications require amplifiers to have different properties of amplification.

An example

Consider a fibre optic communications system with a great distance between transmitter and receiver. Because of the attenuation of light along the length of fibre it is often necessary to compensate for this loss by having 'repeater' circuits at intervals along the fibre. The purpose of these circuits is to detect the incoming light signal, convert it into an electrical signal that can be amplified and then used to drive a secondary light source that

provides a regenerated signal for transmission further down the cable.

A typical sensor for the input of such a repeater is a photodiode. This will produce a current that is proportional to the light intensity. A typical output device might be a light-emitting diode, or perhaps a laser diode, which also needs a current signal to drive it. The amplifier within the repeater must thus take the small current output from the photodiode and amplify it to drive the LED or laser.

Therefore, the amplifier required has to be a **current amplifier**. The requirements for such a circuit are:

Its input should affect the signal current from the photodiode as little as possible,

It should have output circuitry that maximises the current transfer out of the amplifier,

It should have current gain.

To achieve this, it should have **as low an input resistance as possible** and its output should look like an ideal current source (i.e. it should have **as high an output resistance as possible**).

Another example

Consider a communications receiver system. Within the receiver there are circuits that perform functions such as high frequency amplification of the signals, the selection of the required signal (filters) and frequency translation of the signal to a lower frequency (mixers). Because of current limitations on the operating speed of analogue-to-digital converters and other digital circuitry, these functions are normally performed by analogue circuits. However, it is common practice to convert high frequency signals to a much lower frequency (often a few tens of kHz) so that such functions as demodulation or decoding and the final signal processing can be done using digital techniques. DSP (digital signal processing) circuits are widely used to do this.

Typically, DSP chips require input voltage signals of a few volts amplitude. The output signal from the analogue part of the receiver system may be only tens of millivolts in amplitude. An amplifier is therefore required to ‘bridge this gap’.

Therefore, this amplifier has to be a **voltage amplifier**. The requirements for such a circuit are:

Its input should affect the signal voltage from the analogue circuitry as little as possible,

It should have output circuitry that maximises the voltage transfer out of the amplifier,

It should have voltage gain.

To achieve this, it should have **as high an input resistance as possible** and its output should look like an ideal voltage source (i.e. it should have **as low an output resistance as possible**).

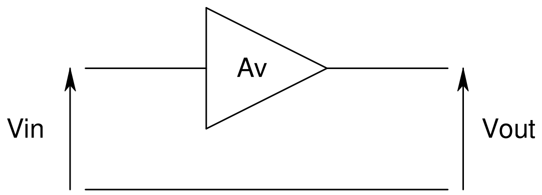
You can imagine that the requirements for the two example amplifiers above will result in completely different circuitry to satisfy them.

The two other types of amplifier also find uses in electronics and communications systems. For example, a field effect transistor amplifier is an example of a **transconductance amplifier** and **transresistance amplifiers** are widely used in audio

mixers and digital-to-analogue converters.

The Voltage Amplifier

A **voltage amplifier** is one to which an input voltage is applied and an output voltage results. The block diagram for such a system is given below.



The **voltage gain** of such an amplifier is given by

$$\text{Voltage gain,} \quad A_v = \frac{V_{out}}{V_{in}}$$

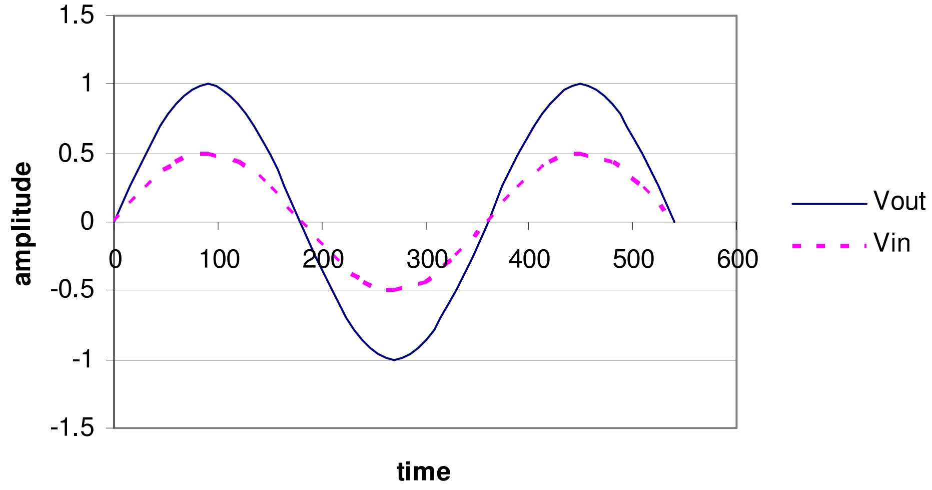
Ideally, connecting a voltage amplifier into a circuit should not affect the input voltage in any way. To achieve this, the input impedance of the amplifier should be as high as possible.

Also, ideally, connecting a load onto the output of the amplifier should not change the output voltage in any way. For this to happen, the output impedance of the amplifier should be as low as possible.

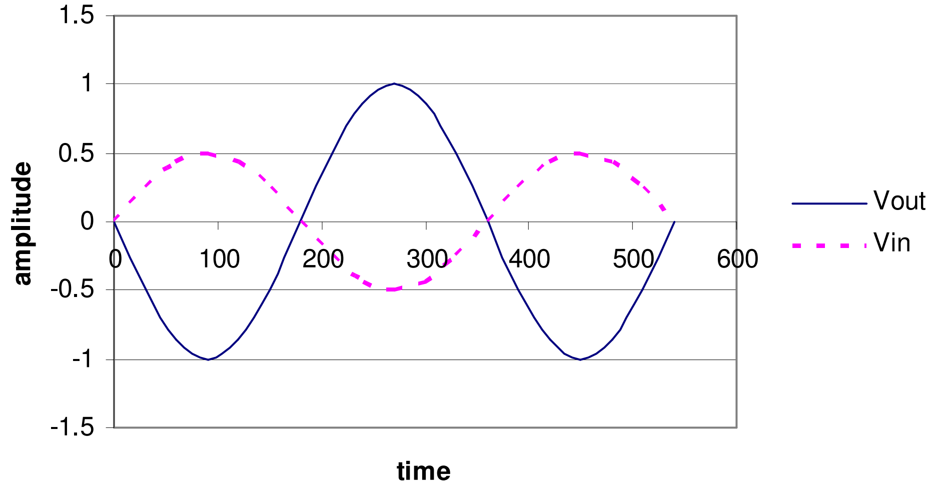
The ideal properties for a voltage amplifier are thus:

Voltage gain	Yes
Input impedance	Very high
Output impedance	Very low

The output voltage may follow the input voltage directly, or it may be inverted in polarity. This is illustrated below:



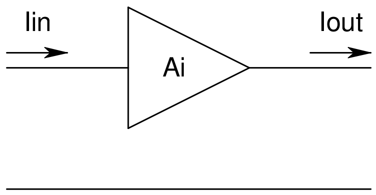
This shows the output following the input directly. The output is said to be **in phase** with the input. Another name for such an amplifier is a **non-inverting amplifier**. Note, also, that the output is amplified with respect to the input.



This shows the output inverted with respect to the input. The output is said to be in **anti-phase** with the input. Another name for such an amplifier is an **inverting amplifier**. Note, also, that the output is amplified with respect to the input.

The Current Amplifier

A **current amplifier** is one to which an input current is applied and an output current results. The block diagram for such a system is given below.



The **current gain** of such an amplifier is given by

$$\text{Current gain,} \quad A_i = \frac{I_{out}}{I_{in}}$$

Ideally, connecting a current amplifier into a circuit should not affect the input current in any way. To achieve this, the input impedance of the amplifier should be as low as possible.

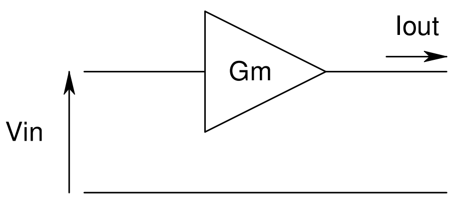
Also, ideally, connecting a load onto the output of the amplifier should not change the output current in any way. For this to happen, the output impedance of the amplifier should be as high as possible.

The ideal properties for a voltage amplifier are thus:

Current gain	Yes
Input impedance	Very low
Output impedance	Very high

The Transconductance Amplifier

A **transconductance amplifier** is one to which an input voltage is applied and an output current results. The block diagram for such a system is given below.



The **transconductance gain** of such an amplifier is given by

$$\text{Transconductance gain,}\quad G_m = \frac{I_{out}}{V_{in}}$$

Ideally, connecting a voltage amplifier into a circuit should not affect the input voltage in any way. To achieve this, the input impedance of the amplifier should be as high as possible.

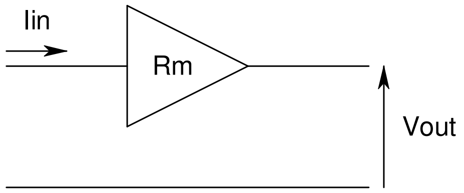
Also, ideally, connecting a load onto the output of the amplifier should not change the output current in any way. For this to happen, the output impedance of the amplifier should be as high as possible.

The ideal properties for a transconductance amplifier are thus:

Transconductance gain	Yes
Input impedance	Very high
Output impedance	Very high

The Transresistance Amplifier

A **transresistance amplifier** is one to which an input current is applied and an output voltage results. The block diagram for such a system is given below.



The **transresistance gain** of such an amplifier is given by

$$\text{Transresistance gain,}\quad R_m = \frac{V_{out}}{I_{in}}$$

Ideally, connecting a transresistance amplifier into a circuit should not affect the input current in any way. To achieve this, the input impedance of the amplifier should be as low as possible.

Also, ideally, connecting a load onto the output of the amplifier should not change the output voltage in any way. For this to happen, the output impedance of the amplifier should be as low as possible.

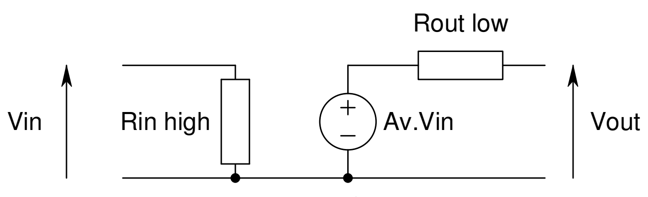
The ideal properties for a transresistance amplifier are thus:

Transresistance gain	Yes
Input impedance	Very low
Output impedance	Very low

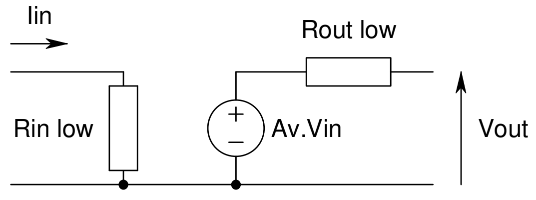
Summary of Amplifier Properties

A summary of the properties of the four types of amplifier, together with an equivalent circuit for each, is given below.

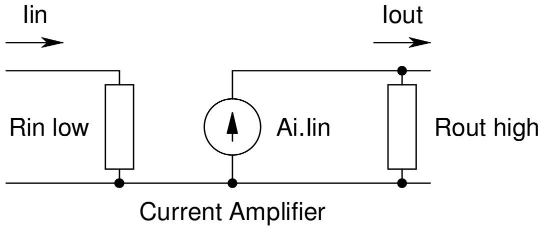
Amplifier Type	**Gain Required**	**Input Resistance**	**Output Resistance**
Voltage Amplifier	Voltage A_V	Very high	Very low
Current Amplifier	Current A_I	Very low	Very high
Transconductance Amplifier	Transconductance G_m	Very high	Very high
Transresistance Amplifier	Transresistance R_m	Very low	Very low

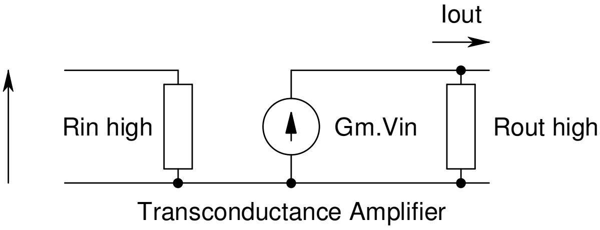


Voltage Amplifier



Transresistance Amplifier


Iin
Iout
Rin low
Ai.Iin
Rout high
Current Amplifier


Iout
Rin high
Gm.Vin
Rout high
Transconductance Amplifier

Classes of Amplifier Operation

RF amplifiers are classified A, AB, B or C according to the phase-angle **(***number of degrees of current flow during each 360-degree RF cycle***)** over which anode- or collector-current flows.

Class A Amplifiers

Class A amplifiers operate over a relatively small portion of a transistor's collector-current (or a vacuum tube's anode-current) range and have continuous collector-current flow throughout each RF cycle. Their efficiency in converting DC-source-power to RF-output-power is poor. DC source power that is not converted to radio frequency output power is dissipated as heat. However, Class A amplifiers do have greater input-to-output waveform linearity (*lower output-signal distortion*) than any other amplifier class. They are most commonly used in small-signal applications where linearity is more important than power efficiency, but also are sometimes used in large-signal applications where the need for extraordinarily high linearity outweighs cost and heat disadvantages associated with poor power efficiency.

Class B Amplifiers

Class B amplifiers have their transistor bases biased near collector-current cut-off, causing collector-current to flow only during approximately 180degrees of each RF cycle. That causes the DC-source-power to RF-output-power efficiency to be much higher than with ClassA amplifiers, but at the cost of severe output cycle waveform distortion. That waveform distortion is greatly reduced in practical designs by using relatively high-Q resonant output circuits to reconstruct full RF cycles.

The effect is the same in principle as pushing a child in a swing through half-swing-cycles and letting the natural oscillatory characteristics of the swing move the child through the other half-cycles. However, low sine-wave distortion results in either case only if the Q of the oscillatory circuit (*the output circuit or the swing*) is sufficiently high. Unless the Q is infinite, which it never can be, the amplitude of one-half cycle will be larger than the other, which is another way of saying there always will be some amount of harmonic energy. (*Coupling an antenna system too tightly to the resonant output circuit of an amplifier will lower its Q, increasing the percentage of harmonic content in the output.*)

Another effective method commonly used to greatly reduce Class B RF amplifier output waveform distortion (*harmonic content*) is to employ two amplifiers operating in “push-pull” such that one conducts on half-cycles where the other is in collector current cut-off. Oscillatory tank circuits are still used in the outputs of Class B push-pull amplifiers to smooth switching transitions from the conduction of one amplifier to the other, and to correct other nonlinearities, but lower-Q circuits can be used for given percentages of harmonic content in the output. (*Output circuits can be loaded more-heavily for given percentages of harmonic output where two amplifiers operate in push-pull.*)

Class AB Amplifiers

As the designation suggests, Class AB amplifiers are compromises between Class A and Class B operation. They are biased so collector current flows less than 360 degrees, but more than 180 degrees, of each RF cycle. Any bias point between those limits can be used, which provides a continuous selection range extending from low distortion, low efficiency on one end to higher distortion, higher efficiency on the other.

Class AB amplifiers are widely used in SSB linear amplifier applications where low distortion and high power-efficiency tend to both be very important. Push-pull Class AB amplifiers are especially attractive in SSB linear amplifier applications because the greater linearity resulting from having one amplifier or the other always conducting makes it possible to bias push-pull Class AB amplifiers closer to the Class B end of the AB scale where the power-efficiency is higher. Alternatively, push-pull Class AB amplifiers can be biased far enough toward the highly linear Class A end of the scale to make broadband operation without resonant output circuits possible in applications where broadband operation or freedom from tuning is more important than power-efficiency.

Class C Amplifiers

Class C amplifiers are biased well beyond cut-off, so that collector current flows less than 180 degrees of each RF cycle. That provides even higher power-efficiency than Class B operation, but with the penalty of even higher input-to-output nonlinearity, making use of relatively high-Q resonant output circuits to restore complete RF sine-wave cycles essential. High amplifying nonlinearity makes them unsuitable to amplify AM, DSB, or SSB signals.

However, most Class C amplifiers can be amplitude modulated with acceptably low distortion by varying the collector voltage, because they generally are operated in the region of collector saturation so that the RF output voltage is very closely dependent upon instantaneous DC collector voltage. They also are commonly used in CW and frequency-shift-keyed radiotelegraph applications and in phase- and frequency-modulated transmitter applications where signal amplitudes remain constant.

Efficiency of Amplifiers

The efficiency of an amplifier is defined as the ratio of the signal power that the amplifier outputs into the load to the total power that the amplifier takes from its power supply.

$$\textbf{Efficiency} = \frac{\textbf{Power into Load}}{\textbf{Power from Supply}}$$

The power into the load is given by

$$P_{out} = \frac{V_{out}^2}{R_L}$$

and the power taken from the supply is given by

$$P_{\sup ply} = V_{dc}.I_c$$

where V_{dc} is the dc voltage across the amplifying device and I_c is the current through the device.

For the amplifier on the workboard, the dc voltage across the device is the supply voltage minus the dc voltage across resistors R3 and R4. However, the value of R3 is only 2Ω, so the voltage across it is very small compared with the supply voltage (12V) and may thus be ignored.

So, the dc voltage across the device is

$$V_{dc} = 12 - V_{R4}$$

The dc current through the device (I_c) is given by the dc voltage across R4 divided by the value of R4 (47Ω).

$$I_c = \frac{V_{R4}}{47}$$

Thus, the efficiency may be calculated.

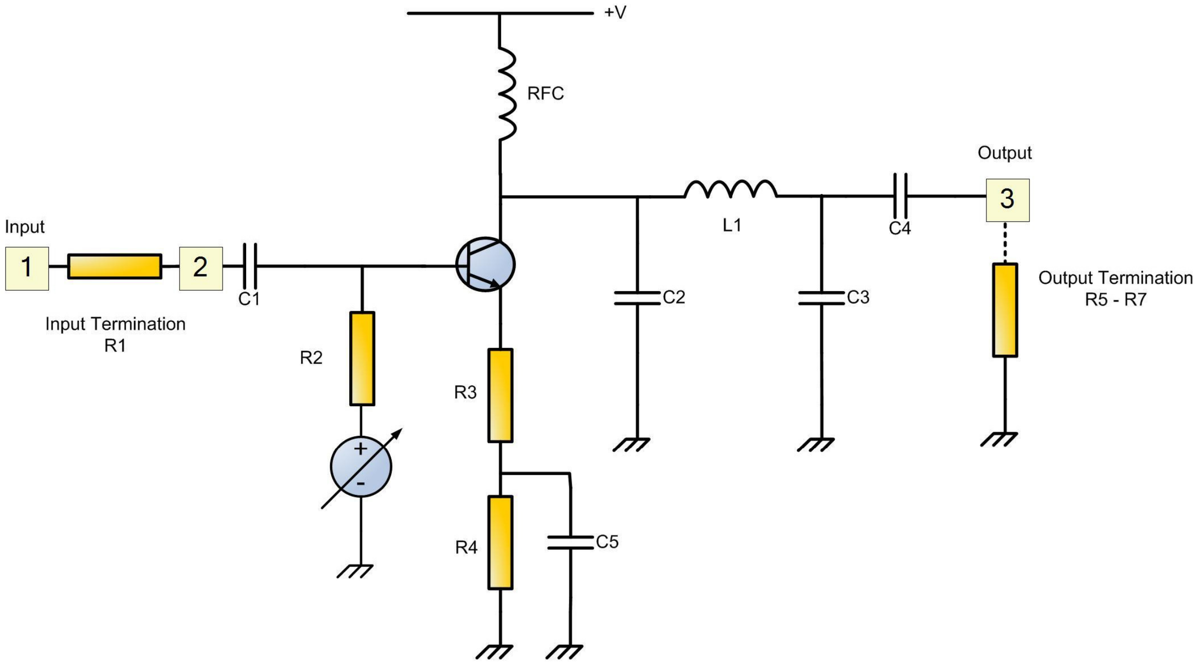
Practical 1: Variation in Gain with Bias Voltage

Objectives and Background

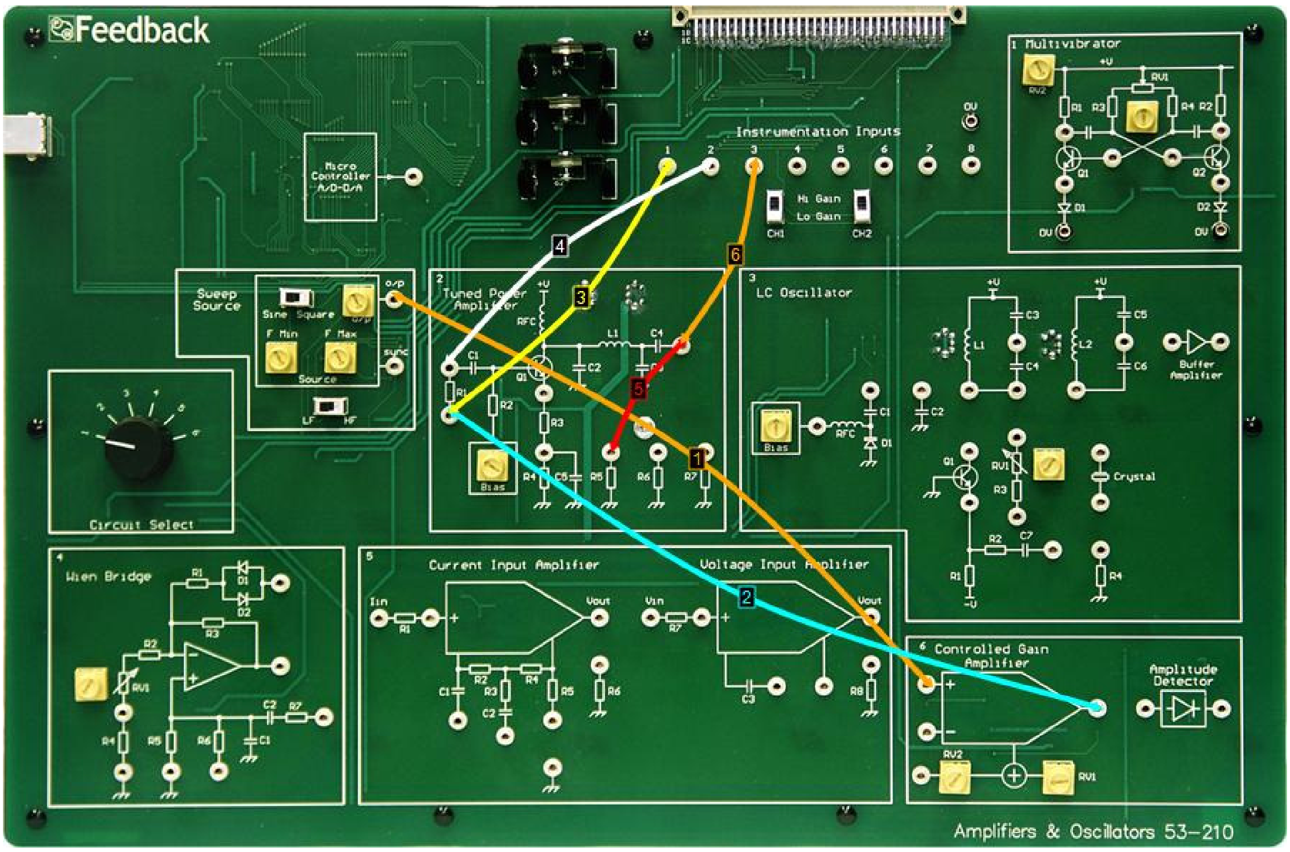
In this Practical you will determine the gain of the amplifier for three settings of bias voltage. The settings will roughly correspond to operating the amplifier first in Class A, then in Classes B and C.

To determine the gain you will adjust the input amplitude to give the same output for the three settings of bias. The load will be kept at 47Ω for each of the settings.

Block Diagram



Make Connections Diagram



Practical 1: Variation in Gain with Bias Voltage

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **Tuned Power Amplifier** circuit block, located towards the centre of the workboard. Set the **Bias** control in this circuit block to its maximum (fully clockwise) position. This sets the amplifier into Class A operation.

Ensure that the **Circuit Select** switch, on the left of the workboard, is set to position **2**.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 switches to **Lo Gain**.

Identify the **Sweep Source** circuit block at the centre-left of the workboard. In this block, set the **Sine**/**Square** switch to Sine and the **LF**/**HF** switch to HF. Set both the **FMin** and **FMax** controls to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Set the **o**/**p** control on the Sweep Source to approximately its 9 o'clock position.

Identify the **Controlled Gain Amplifier** circuit block. This is to be found in the bottom, right-hand corner of the workboard. Set **RV1** in this block to maximum (fully clockwise) and **RV2** to half scale.

Open the oscilloscope and use the FMin control on the Sweep Source to set the frequency to the resonant frequency of the amplifier (vary FMin for maximum output amplitude). Now use the o/p control on the Sweep Source to set the amplifier output amplitude (blue probe on monitor point 3) to approximately 4V pk-pk - you can open the voltmeter to help you set this.

Note the extremely small amplitude of the input signal to give 4V pk-pk output. The gain of the amplifier is very high in Class A.

Now, set the Bias control on the Tuned Power Amplifier to its 9 o'clock position. This approximates to Class B operation of the amplifier.

Readjust the o/p control on the Sweep Source to give an amplifier output signal amplitude of approximately 4V pk-pk. Measure the input signal amplitude at monitor point 2 and estimate the gain of the amplifier. Compare this with that for Class A.

Now, set the Bias control on the Tuned Power Amplifier to its minimum position. This approximates to Class C operation of the amplifier.

Readjust the o/p control on the Sweep Source to give an amplifier output signal amplitude of approximately 4V pk-pk (you may have to slightly readjust the source frequency for resonance to get this output). Measure the input signal amplitude at monitor point 2 and estimate the gain of the amplifier. Compare this with those for Class A and Class B operation.

You should see that the gain of the amplifier decreases as the class of operation goes from Class A towards Class C.

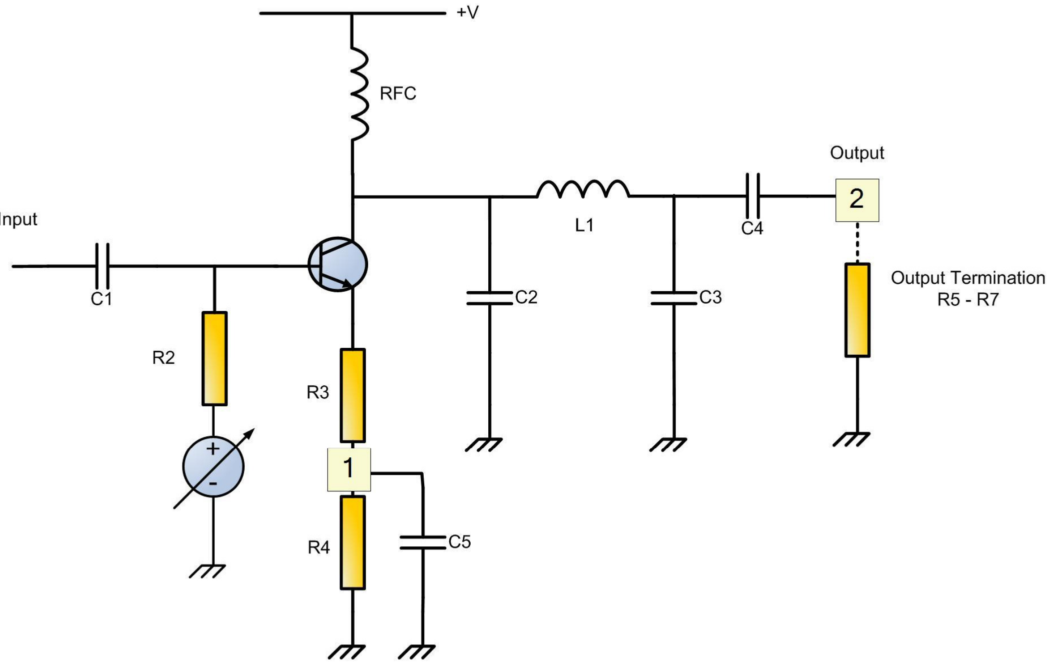
Practical 2: Variation in Efficiency with Bias Voltage

Objectives and Background

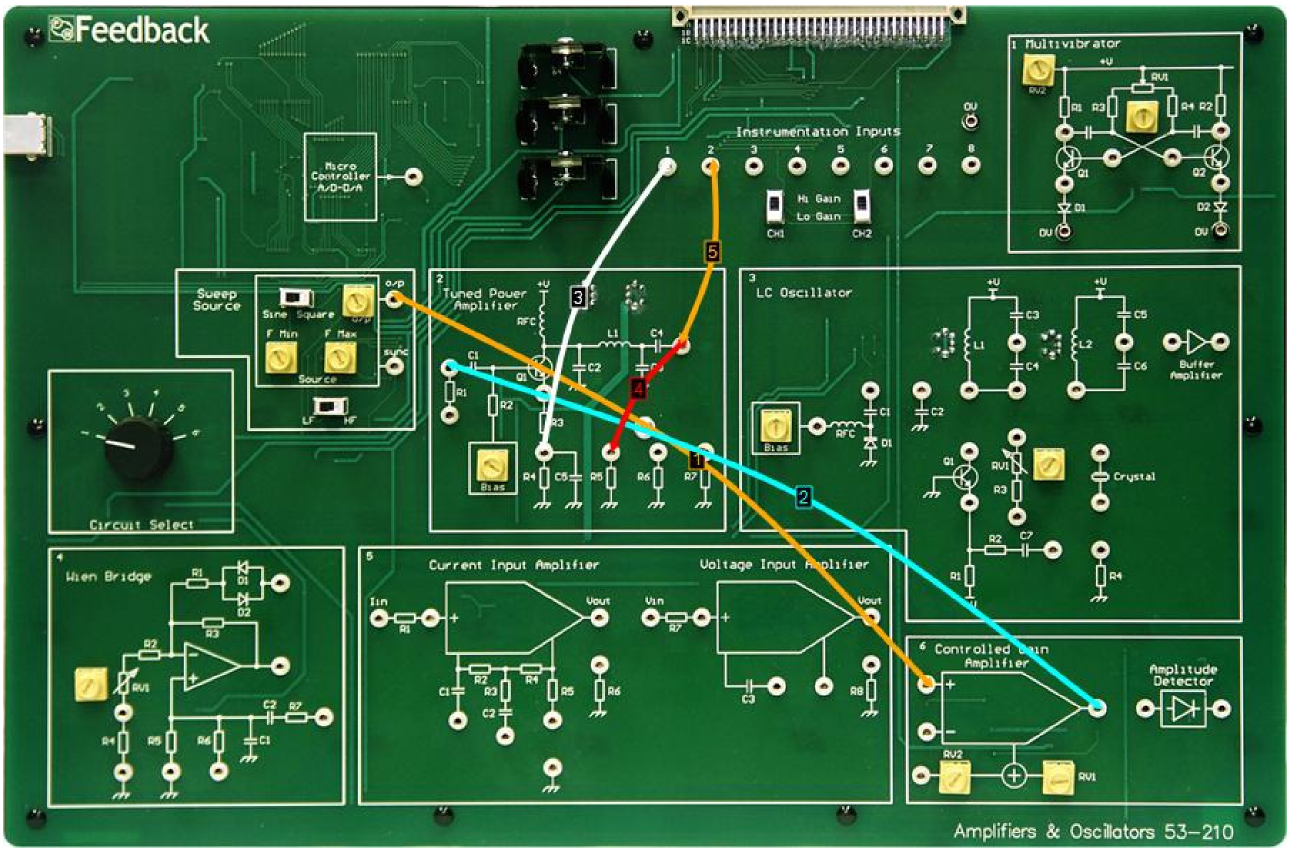
In this Practical you will determine the efficiency of the amplifier for three settings of bias voltage. The settings will roughly correspond to operating the amplifier first in Class A, then in Classes B and C.

To determine the efficiency you will adjust the input amplitude to give the same dc input power for the three settings of bias. You will calculate the signal output powers for each setting and hence estimate the efficiencies for them. The load will be kept at 47Ω for each of the settings.

Block Diagram



Make Connections Diagram



Practical 2: Variation in Efficiency with Bias Voltage

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that the **Circuit Select** switch, on the left of the workboard, is set to position **2**.

Ensure that both **CH1** and **CH2** gain switches are set to **Lo Gain**.

Set the **Bias** control in the **Tuned Power Amplifier** circuit block to its maximum (fully clockwise) position. This sets the amplifier into Class A operation.

In the **Sweep Source** circuit block, set the **Sine**/**Square** switch to Sine and the **LF**/**HF** switch to HF. Set both the **FMin** and **FMax** controls to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Set the **o/p** control on the Sweep Source to approximately its 8 o'clock position.

In the **Controlled Gain Amplifier** circuit block, ensure that **RV1** is set to maximum (fully clockwise) and **RV2** to half scale.

Open the oscilloscope and the spectrum analyser and use the FMin control on the Sweep Source to set the frequency to the resonant frequency of the amplifier (vary FMin for maximum output amplitude). Note: it may be easier to see this on the spectrum analyser than the oscilloscope.

Now use the o/p control on the Sweep Source to set the amplifier output amplitude (blue probe on monitor point 2) so that there is just no second harmonic present in the output spectrum. The amplifier is now operating in Class A whilst giving its maximum linear output.

Open the voltmeter. Measure the output signal amplitude (blue probe on monitor point 2) and also the dc voltage across R4 (yellow probe on monitor point 1). Note: it may be easier to move the yellow probe temporarily to monitor point 2 and switch the voltmeter to **ac pk-pk** to measure the output amplitude.

Calculate the efficiency in Class A using the formulae given in the concept on Efficiency.

Now, set the Bias control in the Tuned Power Amplifier circuit block to its minimum position. This sets the amplifier into Class C operation. Note that there are harmonics present in the output spectrum - the amplifier is no longer linear.

Now adjust the o/p control on the Sweep Source to obtain the same amplifier output amplitude as you had for Class A operation. This will give the same output power into the load in Class C as you had in Class A.

Measure the new dc voltage across R4 and calculate the efficiency for Class C operation.

Compare this efficiency with that for Class A.

Finally, adjust the input amplitude to give maximum output amplitude. Measure the output pk-pk voltage and the dc voltage across R4 and calculate the efficiency for maximum output. Compare this with the other values.

Multvibrator

Objectives

To investigate the factors that determine the frequency of oscillation of a multivibrator

To examine the waveforms associated with a multivibrator and how they are affected by circuit design

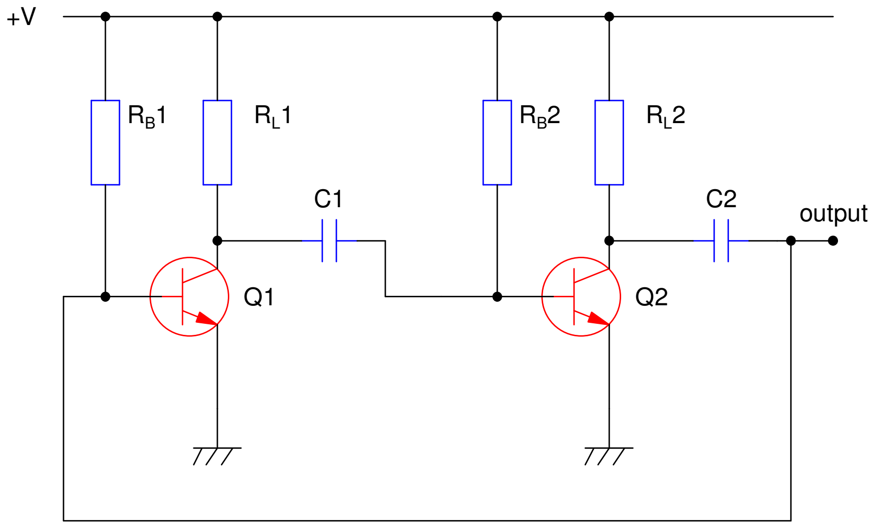
To investigate the effect of leakage in the transistor base-emitter junction and how to prevent it

To investigate variations in mark/space ratio

To examine the spectrum of a square wave

Multivibrator Operation

The Multivibrator circuit comprises two common-emitter transistor switch stages that are cross-coupled by RC networks. The basic circuit is given below:



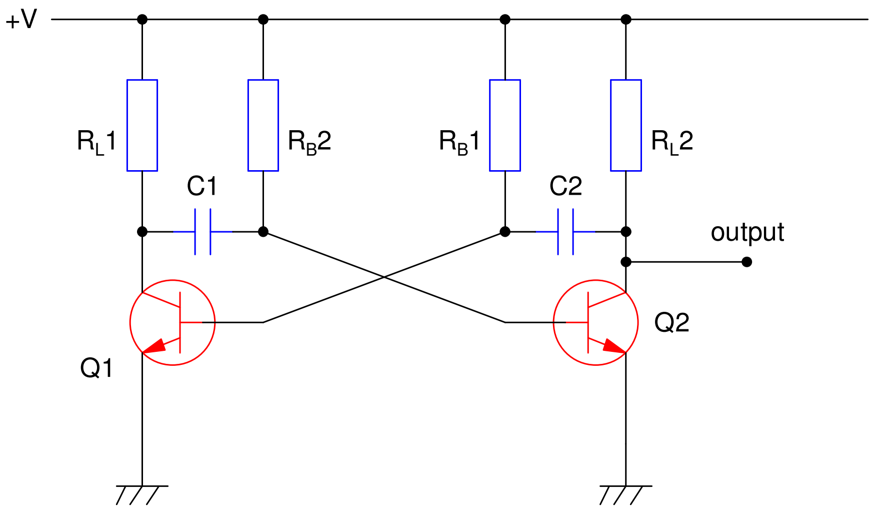
The two transistor stages are inverting switches, so the total phase shift from the input of Q1 to the output of Q2 is 360 degrees. The gain of each stage is high, so the loop gain will be high (> one) – thus the conditions for oscillation are met.

When the supply voltage, +V, is applied, both transistors will try to switch on, due to their R_Bs being connected to +V. However, the two transistors will not have **exactly** the same characteristics, so one of the devices will switch on very slightly before the other. Let us suppose that it is Q2 that switches on first (it doesn't matter which device does). This means that the collector of Q2 switches to 0V. The right-hand end of C2 will follow instantaneously, thus switching Q1 off. The collector of Q2 will therefore switch to +V and there will be no voltage across C1.

However, at the instant that Q2 switches, the voltage at the right-hand end of C2 will start to rise exponentially due to the fact that it is connected to +V via R_B1. At some time – dependant on the time constant of the C2/R_B1 combination – this voltage will rise to the switch-on voltage of Q1. When this occurs, the collector of Q1 will switch to 0V, the right-hand end of C1 will switch to –V and Q2 will switch off. The right-hand end of C1 will then start rising exponentially towards +V (due to it charging via R_B2). When this voltage reaches the switch-on voltage of Q2, this device will then switch on and the whole procedure will start over again.

The resulting voltage at either collector will be a switched waveform with a period dependant on the C1/R_B2 and C2/R_B1 time constants. If the time constants are the same, the waveform will be a square wave with a 1:1 mark/space ratio. Generally, R1 = R2 and C1 = C2, so this is normal.

The common way for the circuit to be drawn is not as shown in the diagram above, but is normally:

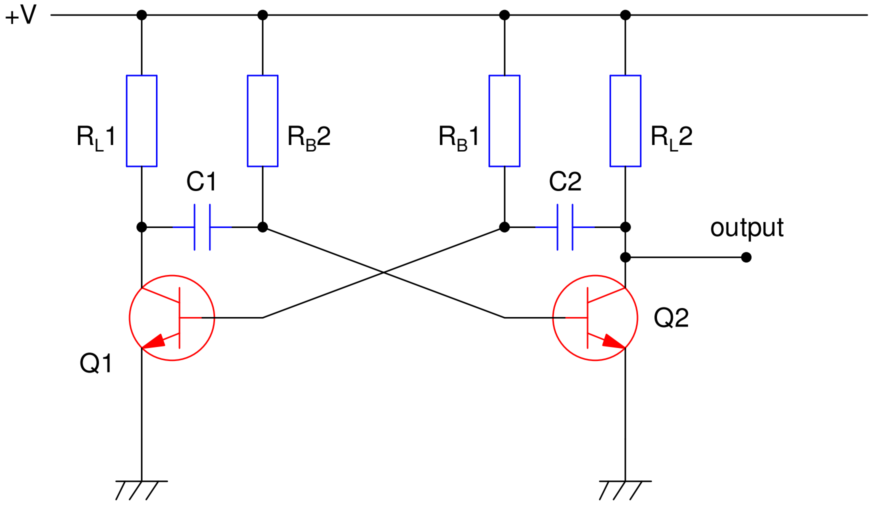


The output can be taken from either collector.

Alternate names that you may come across for a multivibrator are **astable** or **flip-flop**.

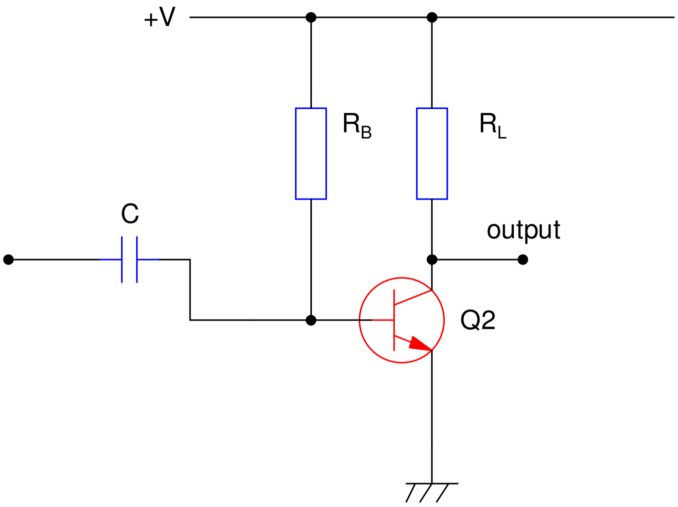
Multivibrator Mathematics

The basic multivibrator circuit is:



Consider the circuitry around Q2.

Let us suppose, for simplicity, that C1 = C2 = C and that $R_B1 = R_B2 = R_B$



When Q1 has just switched on, the left-hand end of C will have dropped from +V to almost 0V.

The right-hand end of C, that had been at $+V_f$ (the V_{BE} 'on' voltage of Q2 - approx +0.6V), will have instantaneously dropped by the same amount; i.e. to $(+V_f – V)$ volts. Normally V is very much greater than V_f, so this can be approximated to –V volts. The voltage across R_B will therefore be 2V volts. Immediately the voltage at the right-hand end of C will start to rise exponentially as C discharges through R_B. The initial current through this resistor will be given by:

$$i = \frac{2V}{R_B}$$

And will decay exponentially with a time constant $C.R_B$

The voltage across R_B is given by:

$$iR_B = 2V.e^{\frac{-t}{CR_B}}$$

So the voltage at the base of Q2 will be:

$$V_b = V - iR_B$$

$$V_b = V - 2V.e^{\frac{-t}{CR_B}}$$

This voltage will rise until the forward switch-on voltage (V_f) of Q2 is reached. This will be at a time, t, given by:

$$V_f = V - 2V.e^{\frac{-t}{CR_B}}$$

i.e., when

$$e^{\frac{-t}{CR_B}} = \frac{V - V_f}{2V}$$

And, as V_f is small compared with V

$$e^{\frac{-t}{CR_B}} = \frac{1}{2}$$

Thus:

$$t = CR_B \log_e 2$$

Giving:

$$\mathbf{t = 0.69CR_B}$$

Remember, this is the time that the circuit will spend in **each** state. The total period of the output square waveform will be double this.

Practical 1: The Basic Multivibrator

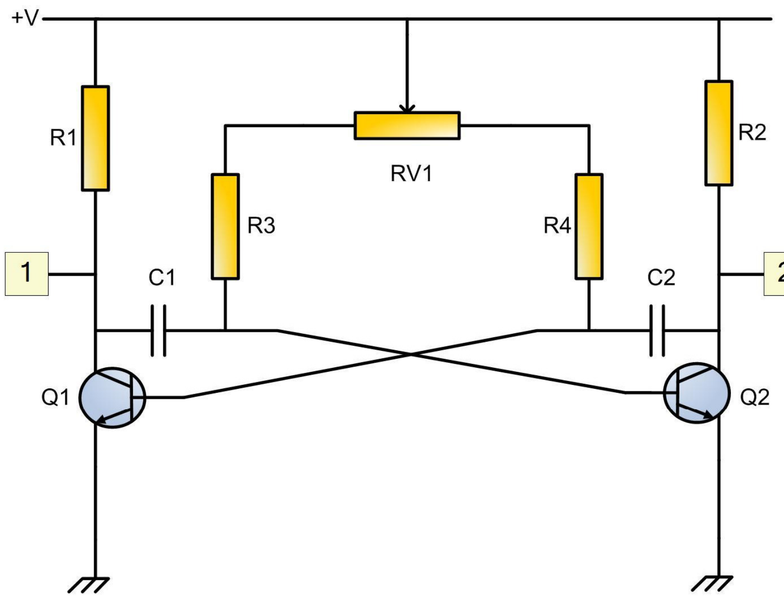
Objectives and Background

In this Practical you will investigate the operation of a basic multivibrator circuit comprising two, common-emitter connected transistors with feedback.

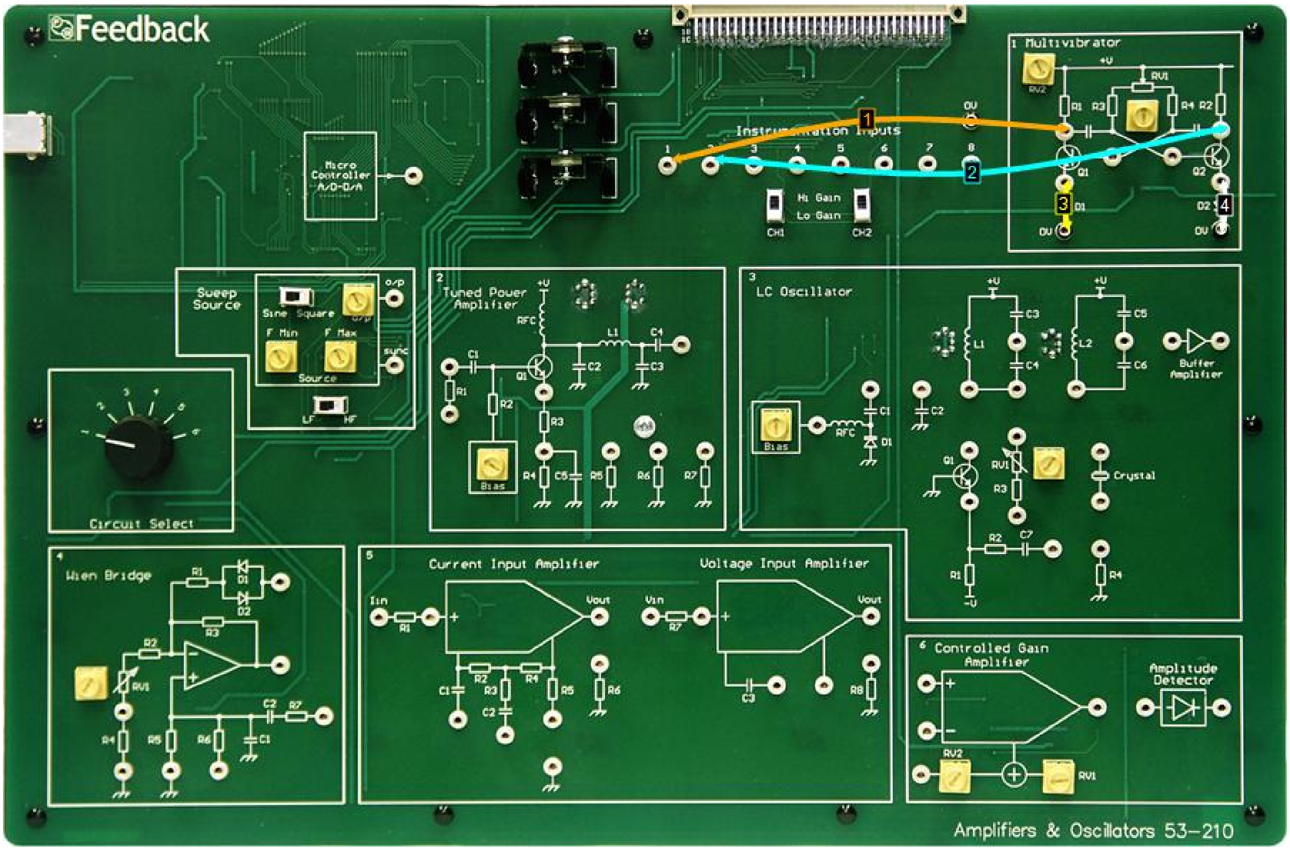
You will see the outputs from the two collectors and their phase relationship.

You will calculate the theoretical frequency of oscillation and compare this with the actual frequency at which the multivibrator operates.

Block Diagram



Make Connections Diagram



Practical 1: The Basic Multivibrator

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **Multivibrator** circuit block, located to the top-right corner of the workboard. In this block, set the **RV1** control to half scale.

Ensure that the **Circuit Select** switch, on the left of the workboard, is set to position **1**.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 switches to **Lo Gain**.

Open the oscilloscope and the frequency counter. Observe the waveforms at the two collectors and the frequency of oscillation.

Notice the relative phases of the two collector waveforms.

Also, measure the mark/space ratio for one of the waveforms. It should be 1:1 if the circuit is perfectly symmetrical. If it is not 1:1, try to think why.

In the circuit on the workboard, R3 and R4 are nominally 68kΩ each, RV1 is nominally 100kΩ and C1 and C2 are both nominally 1nF. Calculate the theoretical frequency of oscillation if RV1 is in mid position and compare this with the measured frequency.

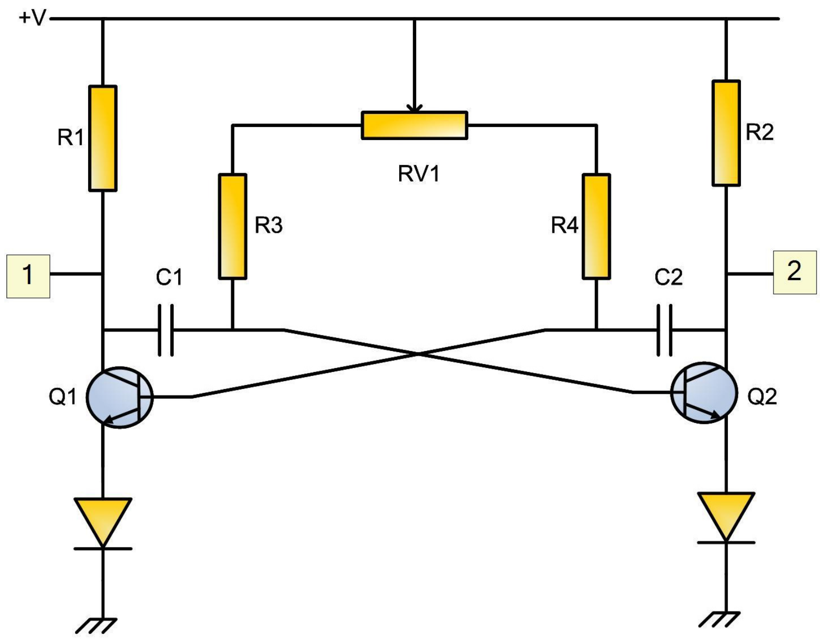
Practical 2: Multivibrator Additions

Objectives and Background

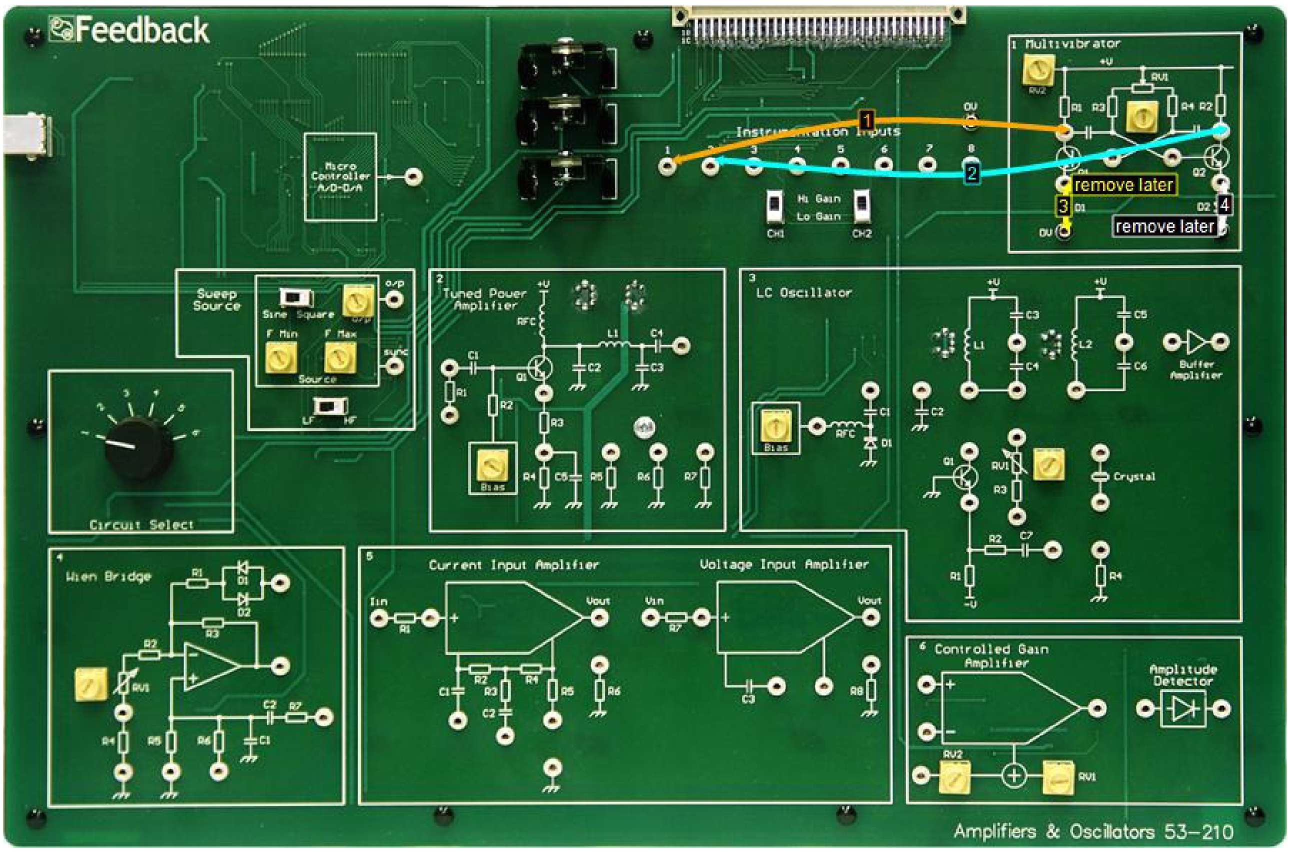
In this Practical you will see the effect of including diodes to protect the transistors has on waveform and frequency.

You will also see how the mark/space ratio of the circuit can be changed and how this affects the operation of the circuit.

Block Diagram



Make Connections Diagram



Practical 2: Multivibrator Additions

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware (the connections are the same as used in Practical 1).

Ensure that the **Circuit Select** switch, on the left of the workboard, is set to position **1**.

Ensure both **CH1** and **CH2** switches to **Lo Gain**.

Set the **RV1** control within the **Multivibrator** circuit block to half scale.

Open the oscilloscope and the frequency counter. Observe the waveforms at the two collectors and the frequency of oscillation. This should not have changed since Practical 1.

Now, remove both the connections that short the two diodes, as shown by connections 3 and 4 on the Make Connections diagram. Notice any change in waveform or frequency.

The diodes that you have just added have introduced an approximate 0.6V difference and thus the theoretical equations given in the Multivibrator Mathematics concept no longer are accurate (they ignore V_{BE} and this diode voltage drop). If your mathematics is up to it, you could re-calculate theoretical equations that include these factors!

Now to see the effect of varying RV1.

Firstly, turn RV1 fully counter-clockwise. Notice the change in mark/space ratio. Also, notice if the frequency of oscillation changes.

Now, turn RV1 fully clockwise and repeat your observations. Explain what you see.

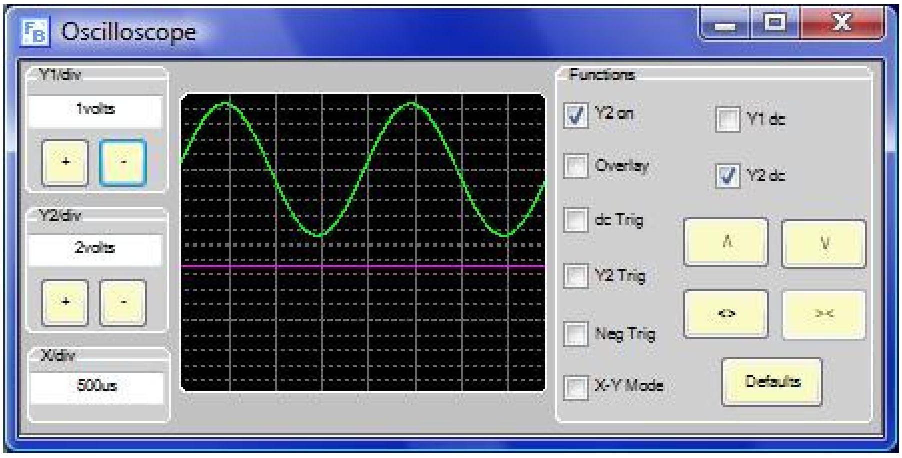
Using the Test Equipment

General Notes

Any of the instruments can be resized and moved at any time using conventional ‘drag-and-drop’ mouse techniques. If you make an instrument small enough then only the display area will be shown; you must increase its size again in order to restore the controls. If you close any of the instruments and open them again they will return to their default settings. Each instrument has a Defaults button which returns the equipment to its default settings (equivalent to closing and re-opening the instrument). If you want to return all the instruments (and any other resource windows) to their default size and position simply click the Auto Position button in the assignment side bar.

Some instruments allow you to place a cursor (by clicking the mouse) at any position on their display; the cursor reveals information regarding the point at which it is located. You will have to reactivate this cursor each time you change the settings, size or position of the instrument.

The Oscilloscope



The Discovery oscilloscope has many of the functions that you would find on a conventional or computer-driven scope. Its fundamental purpose is to show varying waveforms plotted against time. It is a dual trace scope, which means that it can display two separate waveforms at the same time.

The Y (voltage) axis is set to a default value by the practical for each channel, but you may change it by using the + button for more volts/div and the - button for less volts/div.
Either only one channel can be displayed or both channels. The Y2 Show tick box determines whether the second channel is shown. In two-channel mode, if the Overlay box is ticked, the two traces are superimposed on the same scale as for one trace. If Overlay is not ticked the display area is divided into two and each trace is displayed half-

size. The Y1 dc and Y2 dc tick-boxes determine if the inputs are dc coupled or not (ac coupled). If the signal has a large dc offset then ac coupling can be useful.

The X (time) axis is set to a default value by the practical but you may change it by using the ^ button for a faster timebase and the v button for a slower timebase. The <> and >< buttons provide a means of further expanding the trace if the highest, or lowest, timebase is in use. If you have the X scale expanded and select a lower timebase speed then the X scale automatically returns to its default setting.

An anti-alias feature automatically switches the time-base speed up if you select a rate that may produce a misleading display due to aliasing. If this feature has increased the timebase rate then the ^ button is coloured red.

The oscilloscope can also be operated in X-Y mode, where data from channel 1 is in the vertical axis and data from channel 2 is in the horizontal axis. Because the oscilloscope is a digital sampling scope, in X-Y mode the time base settings are still relevant and determine the sampling rate for both channels. Also in X-Y mode the traces have persistence and stay on the screen longer than one trace refresh.

Note that you can switch off the anti-aliasing feature from the main laboratory screen.

Triggering takes place when the selected trace crosses the zero volt level. If the Y2 Trig box is ticked, then the trigger source is Channel 2. Otherwise, Channel 1 is used. The Neg trig box enables only negative transitions to trigger the scope. Normally only positive ones do.

If the signal has a large dc offset, ac triggering can be useful.

You can return to the default settings by pressing the Default button. The Auto Position button on the Discovery laboratory window moves **all** the test instruments back to their default positions and sizes on the screen but does not affect their settings.

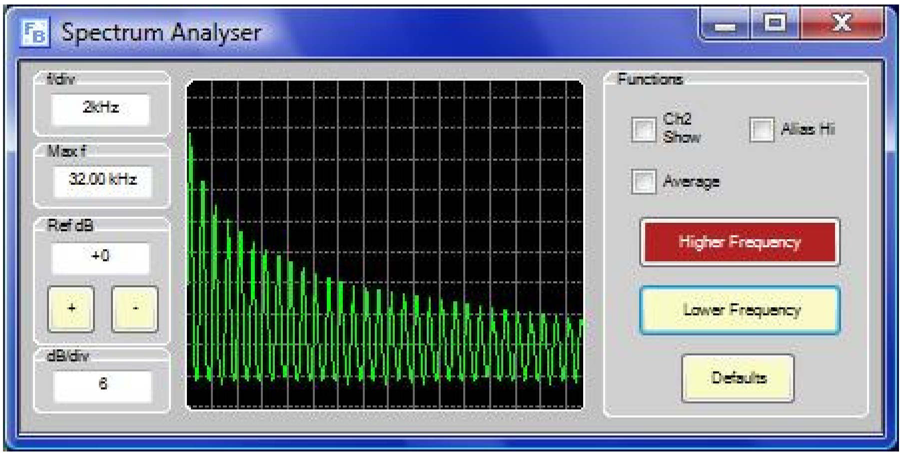
A cursor is available to make more accurate measurements. Left click on the display area to activate it. The green cursor can be moved to anywhere on a waveform. Move the mouse away and back into it to allow a tool-tip window to open with the measurement data displayed for that point.

You have to reactivate the cursor if you change the settings, size or position of the oscilloscope.

By right clicking on the display an options box appears. The options available are:

Print Display – Sends image to the default printer.
Export Display to File – Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.
Export Display to File (reverse colours) - Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.

The Spectrum Analyser



The spectrum analyser enables you to look at signals in the frequency domain. In common with many modern test instruments, it uses DSP to transform time domain data into frequency domain data. The mathematics to do this is called a Fourier transform.

The Y (amplitude) scale is calibrated in Decibels relative to an arbitrary dotted line near to the top of the screen. The dB scale is linear and the number of dB per division is shown in the box. The Y (amplitude) axis is set to a default value by the practical, but you may change it by using the + button or the - button to change the Ref dB value higher or lower. The minimum level that you can see is determined by the assignment, and ultimately by the noise in the system.

The analyser has the capability of showing two channels at the same time. Click Ch2 Show button to show channel 2 as well as channel 1.

The X (frequency) axis is calibrated in MHz, kHz or Hz per division, as appropriate. The default scale is set by the practical but you may change it by using the Higher Frequency and Lower Frequency buttons.

The anti-alias feature will operate if you try to set the frequency too low. The Higher Frequency button is shown red if this feature has increased the frequency. Note that if a new frequency component appears such as noise, the anti-alias feature may operate suddenly. The Alias Hi tick-box allows you to increase the threshold at which the anti-alias feature operates. This allows signals to be examined that have larger amounts of harmonic content. The default setting for this is off.

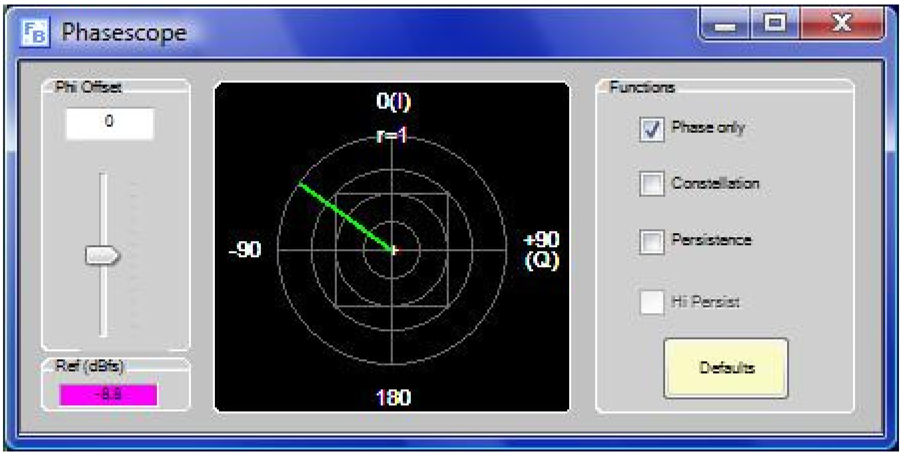
A cursor is available to make more accurate measurements. Left click on the display area to activate it. The green cursor can be moved to anywhere on a waveform. Move the mouse away and back into it to allow a tool-tip window to open with the measurement data displayed for that point.

You will have to reactivate the cursor if you change the settings, size or position of the spectrum analyser.

By right clicking on the display an options box appears. The options available are:

Print Display – Sends image to the default printer.
Export Display to File – Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.
Export Display to File (reverse colours) - Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.

The Phasescope



The Phasescope is a special instrument that compares two signals in phase and amplitude (magnitude). The two signals are referred to as the reference and the input. The display is in polar format, i.e. the phase is in the form of a circle and the amplitude as the radius. The use of a circle is possible because phase is a continuous function repeating every 360 degrees. The display can be seen as Polar, as the one orthogonal axis represents the real component and other the imaginary part. The convention here is that the real axis is the X axis, which means that zero degrees is straight up or at 12 on a clock face. +90 degrees is at 3 on the clock face and –90 at 9. It is important to note that in terms of phase +180 degrees is the same as –180 degrees.

The radius scale has one circle at radius = 1 (the outermost circle) i.e. the two signals are of the same amplitude. Further inner circles are at 0.707, 0.5 and 0.25.

The circle at 0.5 has a square associated with it, the corners of which are at 0.707. This represents the case when two orthogonal vectors of amplitude = 0.5 are added.

In some cases only the phase is of interest so, if you click the Phase Only box, the radius is set to 1.

The conventional display is that of a vector i.e. a line joining the point to the centre. However, in some cases it is much easier to interpret the display if only a point is drawn. Where the amplitude and phase is varying between discrete values they are shown as a pattern of dots resembling stars, hence the term constellation display. This mode can be selected by ticking the Constellation box. In constellation mode, the persistence of the display can be varied. By selecting the Persistence tick box, traces stay on the screen for a number of trace refreshes before being removed. By selecting Hi Persist this time is extended.

If the two signals are of different frequencies the result is a continuously rotating vector, rotating at a rate equal to the difference in frequency. The direction depends on the sign of the frequency difference. If the rate is fairly fast, the instrument may only be able to show a limited number of discrete values.

In many cases the reference input will not be at exactly zero degrees with respect to the theoretical zero degrees of the input signal. This causes the display to be rotated. In some cases this may be important to know, but where it is not the Phi Offset control gives the ability to rotate the display for easier interpretation.

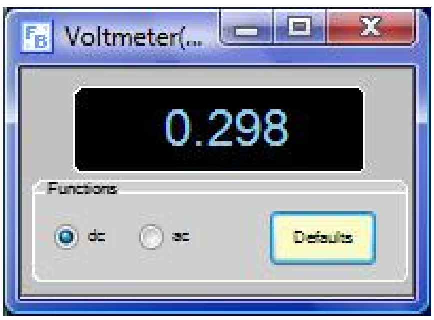
The coloured indicator (Ref Ch) to the top left of the display tells you which probe is being used as the reference channel.

A cursor is available to enable more accurate measurement. Click the display to use it.

By right clicking on the display an options box appears. The options available are:

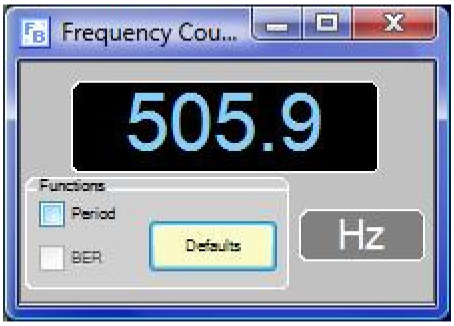
Print Display – Sends image to the default printer.
Export Display to File – Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.
Export Display to File (reverse colours) - Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.

The Voltmeter



The meter is simply an ac and dc voltmeter that displays the value in digital form. It can be used in ac mode by clicking ac p-p, in which case the value represents the peak to peak value. If the waveform has a high crest factor the results can be slightly surprising. In dc mode, if there is an ac component present, the average value is displayed.

The Frequency Counter



This has the facilities of a conventional frequency meter/counter. It will display in either frequency or time. If the input amplitude is too low a warning message will be displayed.

Like all frequency counters, it can produce misleading results if the waveform is complex or contains many frequencies.

The Gain Phase Analyser (GPA)

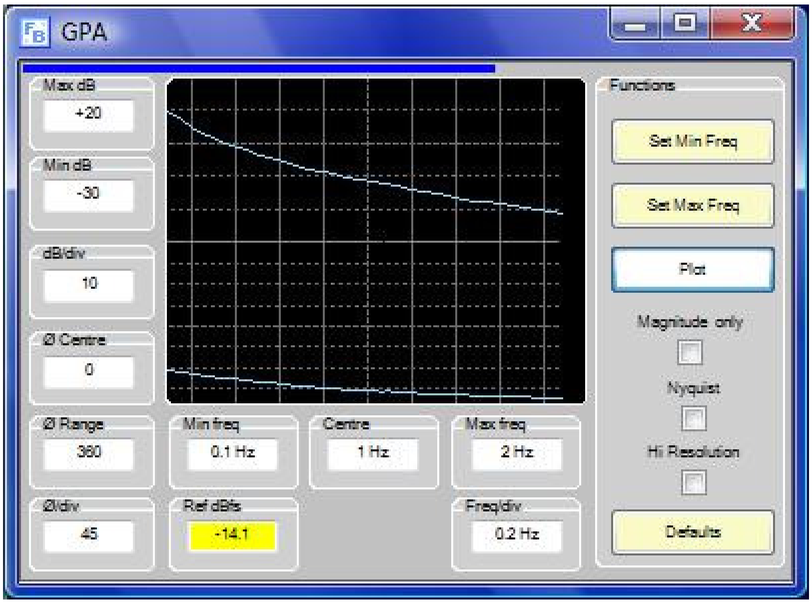
The GPA displays graphically the gain and phase characteristics between a selectable range of frequencies, either as Bode plots (the default mode) or as a Nyquist plot.

Whenever the GPA is opened, the Set Min Freq button on the GPA is initially selected. The minimum frequency that you set in the circuit whilst this button is selected will represent the low end of the range to be plotted. After setting the minimum frequency, select the Set Max Freq button and adjust the maximum frequency in the circuit to the high end of the range. When either the Set Min Freq or Set Max Freq buttons are selected, the frequency will be shown numerically on the GPA display. Finally, select the Plot button to plot the gain and phase between the frequencies you have chosen. Note that the GPA takes time to plot at low frequencies.

You can verify (and change) the minimum or maximum frequencies at any time by selecting the Set Min Freq or Set Max Freq buttons as required (and adjusting the low or high frequency if desired). You will then need to select the Plot button again. Note that only one of these three buttons can be selected at any time.

Tick the Hi Resolution box to get a better frequency resolution of the plot, although plotting will take longer.

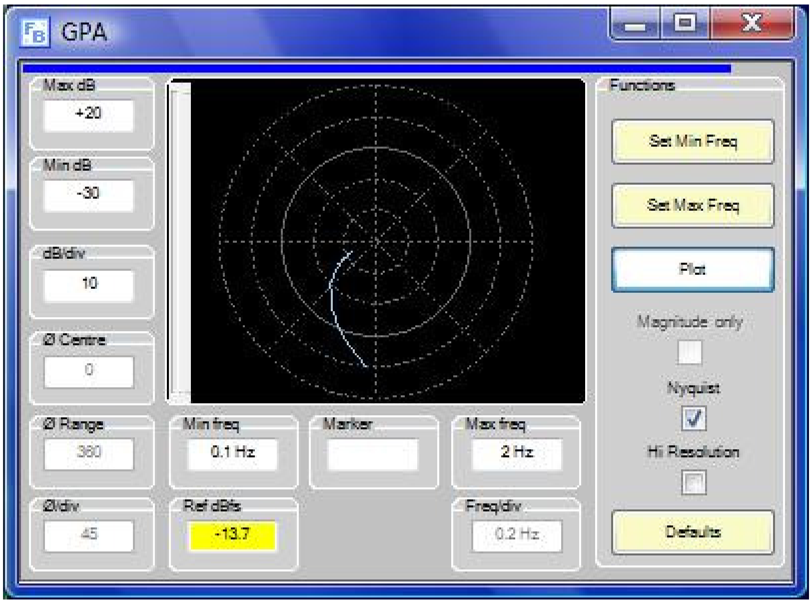
Bode plots are the most widely used means of displaying and communicating frequency response information. Bode diagrams are presented as two separate graphs: one showing magnitude and one showing phase, both plotted against frequency. Because the axes are logarithmic, they condense a wide range of frequencies (horizontal axis) and a wide range of gains (vertical axis) into the graphical area. In Bode plots, commonly encountered frequency responses have a shape that is simple and easy to recognise.



A Bode diagram shows two traces, representing the magnitude and phase, from which you can see variations with frequency. You are able to measure the gain and phase at different frequencies by clicking the cursor on the GPA display. It is much easier to use the GPA than the oscilloscope for this purpose.

Tick the Magnitude only box if you prefer to display the gain trace without the phase trace. This box is disabled when you select a Nyquist plot.

The Nyquist plot of a system is simply the polar representation of the Bode plots. This plot combines the magnitude and phase on a single graph, with the frequency as a parameter along the curve. Nyquist plots are particularly helpful for stability analysis in control system design.



On the GPA, tick the Nyquist box to create a Nyquist plot (you should already have set the minimum and maximum frequencies as described above). Use the vertical slider bar at the left-hand side of the GPA display to move the cursor to any point along the plotted curve. Then read off the frequency, magnitude and phase by hovering the mouse over that cursor. The convention here is that the zero degrees radial is at 3 on a clock face.

Untick the Nyquist box to return to Bode plot mode. You can switch between Bode and Nyquist modes at any time.

By right clicking on the display an options box appears. The options available are:

Print Display – Sends image to the default printer.
Export Display to File – Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.
Export Display to File (reverse colours) - Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.

Discovery System Help

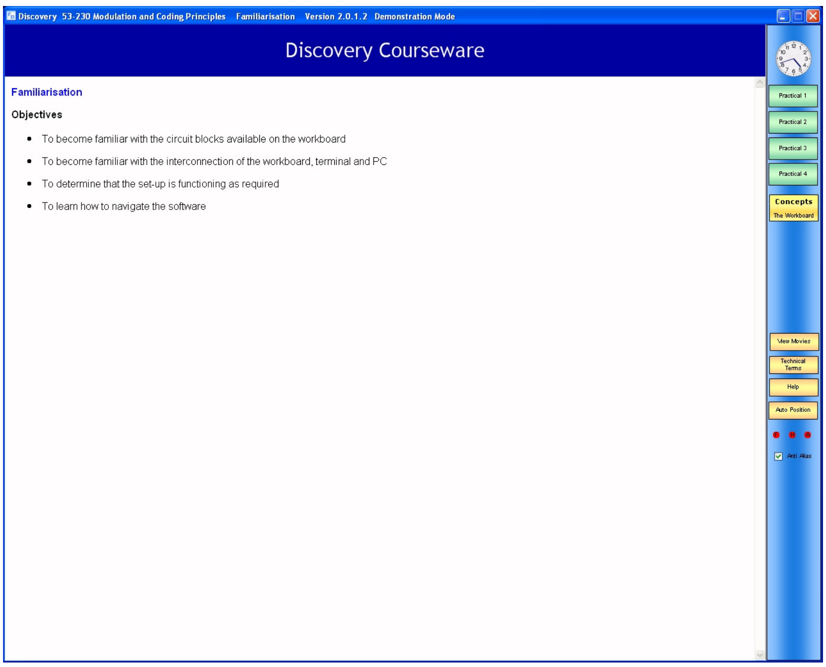
Although the Discovery environment is very easy to operate, these notes will help you use all its facilities more quickly.

If there is a demonstration assignment, slider controls in the software perform functions that would normally be performed on the hardware. In normal assignments, if the any of hardware systems fail to initialise the system reverts to demonstration mode. This means that none of the test equipment is available.

The Assignment Window

The assignment window opens when an assignment is launched. If you are reading this you have already found the help button in the side bar of the assignment window!

The assignment window consists of a title bar across the top, an assignment side bar at the right-hand edge, and the main working area. By default, the overall assignment objectives are initially shown in the main working area whenever an assignment is opened. The assignment window occupies the entire screen space and it cannot be resized (but it can be moved by ‘dragging’ the title bar, and it can be minimised to the task bar). The title bar includes the name of the selected assignment. The side bar contains the practicals and any additional resources that are relevant for the selected assignment. The side bar cannot be repositioned from the right-hand edge of the assignment window. An example of an assignment window is shown below.



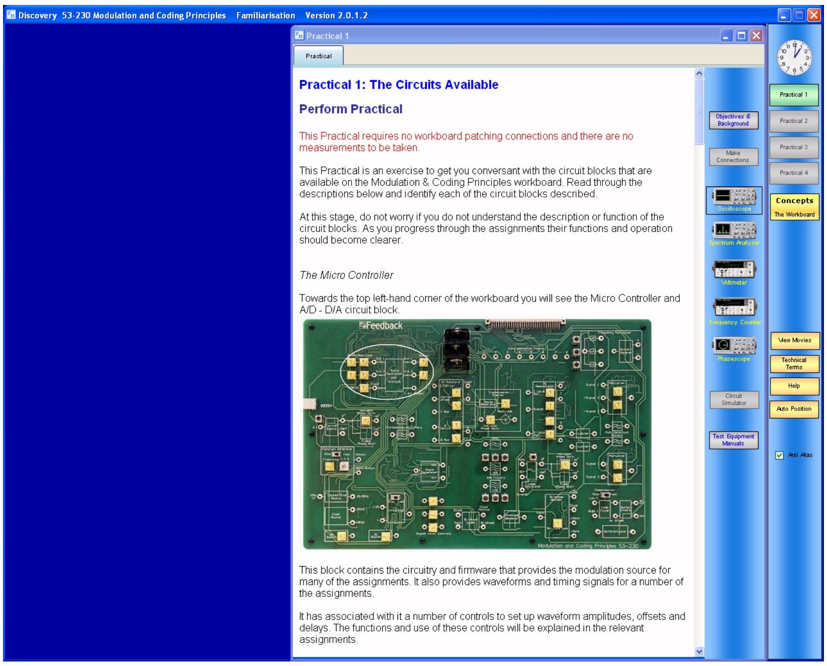
The precise appearance of the assignment window will depend on the ‘skin’ that has been selected by your tutor. However, the behaviour of each of the buttons and icons will remain the same, irrespective of this.

The clock (if you have one active) at the top of the side bar retrieves its time from the computer system clock. By double clicking on the clock turns it into a stop watch. To start the stop watch single click on the clock, click again to stop the stop watch. Double clicking again will return it to the clock function.

There are a number of resource buttons available in the assignment side bar. These are relevant to the selected assignment. In general, the resources available will vary with the assignment. For example, some assignments have video clips and some do not. However, the Technical Terms, Help and Auto Position buttons have identical functionality in every assignment. You can click on any resource in any order, close them again, or minimise them to suit the way you work.

Practicals are listed in numerical order in the side bar. When you hover the mouse over a practical button, its proper title will briefly be shown in a pop-up tool-tip. There can be up to four practicals in any assignment. You can have only one practical window open at any time.

To perform a practical, left-click on its button in the assignment side bar. The assignment objectives, if shown in the main working area, will close, and the selected practical will appear in its own window initially on the right-hand side of the main working area, as shown below. You can move and resize the practical window as desired (even beyond the assignment window). However, its default size and position is designed to allow the test equipment to be displayed down the left-hand side of the main working area without overlapping the instructions for the practical.



Again, the precise appearance of the practical window can be determined by your tutor but the behaviour of each of the buttons and icons will remain the same, irrespective of this. Whatever it looks like, the practical window should have icons for the test equipment, together with buttons for Objectives & Background, Make Connections, Circuit Simulator and Test Equipment Manuals. These resources are found in side bar, located on the right-hand edge of the practical window. The resources will depend on which practical you have selected. Therefore not all the resources are available in every practical. If a resource is unavailable, it will be shown greyed out. To open any resource, left-click on its icon or button. Note that when you close a practical window, any resources that you have opened will close. You may open any resource at any time, provided it available during the practical. The Circuit Simulator will only be available if you have one loaded.

Note that if the hardware is switched off, unavailable, or its software driver is not installed, all the test equipment is disabled. However, you can open any other window. If you switch on the hardware it will be necessary to close the assignment window and open it again to enable the test equipment.

Resource Windows

These are windows may be moved, resized and scrolled. You may minimise or maximise them. The system defaults to ‘Auto Position’, which means that as you open each resource window it places it in a convenient position. Most resource windows initially place themselves inside the practical window, selectable using tabs. Each one lays over the previous one. You can select which one is on top by clicking the tab at the top of the practical window. You can see how many windows you have open from the number of tabs. If you want to see several windows at once then drag them out of the practical window to where you wish on the screen. If you close a window it disappears from the resources tab bar.

If you want to return all the windows to their default size and position simply click the Auto Position button in the assignment side bar.

Make Connections Window

This movable and resizable window shows the wire connections (2mm patch leads) you need to make on the hardware to make a practical work. Note that some of the wires connect the monitoring points into the data acquisition switch matrix. If this is not done correctly the monitoring points on the practical diagram will not correspond with those on the hardware. The window opens with no connections shown. You can show the connections one by one by clicking the Show Next button or simply pressing the space bar on the keyboard. If you want to remove the connections and start again click the Start

Again button. The Show Function button toggles the appearance of the block circuit diagram associated with the practical.

Test Equipment

The test instruments will auto-place themselves on the left of the screen at a default size. You may move or resize any instrument at any time. Note that below a useable size only the screen of the instrument will be shown, without the adjustment controls. Each piece of test equipment will launch with default settings. You may change these settings at any time. There is an auto anti-alias feature that prevents you setting time-base or frequency settings that may give misleading displays. If auto anti-alias has operated the button turns red. You can turn off the anti-aliasing feature, but you should be aware that it may result in misleading displays.

You may return to the default settings by pressing the Default button on each piece of test equipment. If you wish to return all the equipment to their original positions on the left of the screen click Auto Position on the side bar of the assignment window.

Note that if you close a piece of test equipment and open it again it returns to its default position and settings.

If you want more information on how a piece of test equipment works and how to interpret the displays, see the Test Equipment Manuals resource in the practical side bar.

On slower computers it may be noticeable that the refresh rate of each instrument is reduced if all the instruments are open at once. If this is an issue then only have open the instrument(s) you actually need to use.

Test Equipment Cursors

If you left click on the display of a piece of test equipment that has a screen, a green cursor marker will appear where you have clicked. Click to move the cursor to the part of the trace that you wish to measure. If you then move the mouse into the cursor a tool-tip will appear displaying the values representing that position. Note if you resize or change settings any current cursor will be removed.

Practical Window

This window contains the instructions for performing the practical, as well as a block, or circuit, diagram showing the circuit parts of the hardware board involved in the practical. On the diagram are the monitoring points that you use to explore how the system works and to make measurements. The horizontal divider bar between the instructions and the diagram can be moved up and down if you want the relative size of the practical instruction window to diagram to be different. Note that the aspect ratio of the diagram is fixed.

Information Buttons on Practical Diagrams

On many of the symbols on the diagram you will find a button that gives access to new windows that provide more information on the circuit that the symbol represents. Note that these windows are "modal", which means that you can have only one open at a time and you must close it before continuing with anything else.

A Further Information point looks like this 

Probes

The practical diagram has probes on it, which start in default positions. These determine where on the hardware the signals are being monitored.

Selecting and Moving the Probes

Probes are indicated by the coloured icons like this .

If this probe is the *selected probe* it then looks like this  (notice the black top to the probe). You select a probe by left clicking on it.

Monitor points look like this 

If you place the mouse over a monitor point a tool-tip will show a description of what signal it is.

You can move the selected probe by simply clicking on the required monitor point. If you want to move the probe again you do not have to re-select it. To change which probe is selected click on the probe you want to select.

You can also move a probe by the normal 'drag-and-drop' method, common to 'Windows' programs.

Probes and Test Equipment Traces

The association between probes and traces displayed on the test equipment is by colour. Data from the blue probe is displayed as a blue trace. Yellow, orange and green probes and traces operate in a similar way. Which piece of test equipment is allocated to which probe is defined by the practical.

Note that the phasescope shows the relative phase and magnitude of the signal on its input probe using another probe as the reference. The reference probe colour is indicated by the coloured square to the top left corner of the phasescope display.

Practical Buttons

On some practicals there are buttons at the bottom of the diagram that select some parameter in the practical. These can be single buttons or in groups. Only one of each button in a group may be selected at one time.

Slider Controls

Where slider controls are used you may find you can get finer control by clicking on it and then using the up and down arrow keys on your keyboard.

More information on Signal Sources

A signal source is usually some kind of generator that produces a signal output with no signal input. An oscillator is an example of a signal source. Of course, the amplitude, frequency and waveshape may take many values, or it could be simply a constant dc voltage. Usually, we differentiate between a dc source that is used as a power supply and that used for a signal in a circuit.

Sources may be completely autonomous, i.e. they have no inputs at all, simply an output. However, many sources have inputs that control an output parameter such as amplitude or frequency. These are called control inputs. Some may have synchronising inputs that allow the output phase or frequency to be locked to an input signal. To be regarded as a true source the output should continue when all the control signals are removed.